HEIDENHAIN

TNC 410
TNC 426
TNC 430

NC Software
286 060-xx
286 080-xx
280 476-xx
280 477-xx

User's Manual
ISO Programming

English (en)
8/2002

Controls on the visual display unit

	Split screen layout
	Switch between machining or programming modes
	Soft keys for selecting functions in screen
◁ ▷	Switching the soft-key rows
	Changing the screen settings (only BC 120)

Typewriter keyboard for entering letters and symbols

Q W E R T Y	File names Comments
G F S T M	ISO programs

Machine operating modes

	MANUAL OPERATION
	ELECTRONIC HANDWHEEL
	POSITIONING WITH MDI
	PROGRAM RUN, SINGLE BLOCK
	PROGRAM RUN, FULL SEQUENCE

Programming modes

	PROGRAMMING AND EDITING
	TEST RUN

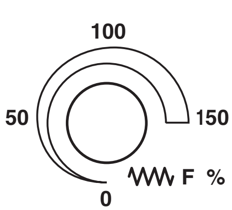
Program/file management, TNC functions

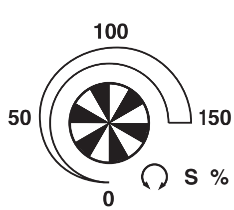
PGM MGT	Select or delete programs and files External data transfer
PGM CALL	Enter program call in a program
MOD	MOD functions
HELP	Display help texts for NC error messages
CALC	Pocket calculator

Moving the highlight, going directly to blocks, cycles and parameter functions

↑ ↓ ← →	Move highlight
GOTO □	Go directly to blocks, cycles and parameter functions

Override control knobs for feed rate/spindle speed





Programming path movements

APPR DEP	Approach/depart contour
FK	FK free contour programming
L	Straight line
CC	Circle center/pole for polar coordinates
C	Circular arc with center
CR	Circular arc with radius
CT	Circular arc with tangential connection
CHF	Chamfer
RND	Corner rounding

Tool functions

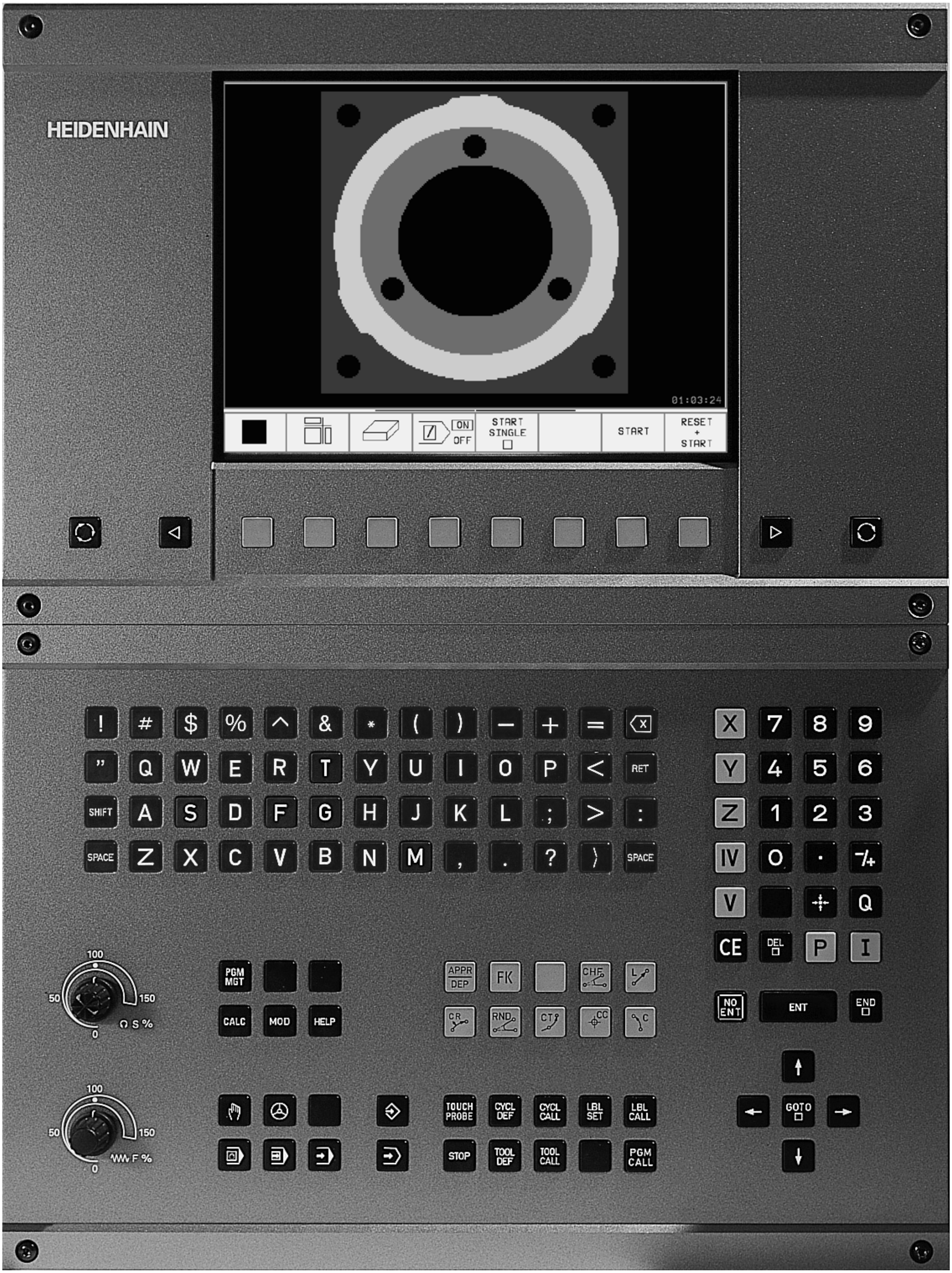
TOOL DEF, TOOL CALL	Enter and call tool length and radius

Cycles, subprograms and program section repeats

CYCL DEF, CYCL CALL	Define and call cycles
LBL SET, LBL CALL	Enter and call labels for subprogramming and program section repeats
STOP	Program stop in a program
TOUCH PROBE	Enter touch probe functions in a program

Coordinate axes and numbers: Entering and editing

X . . . V	Select coordinate axes or enter them into the program
0 . . . 9	Numbers
•	Decimal point
-/+	Change arithmetic sign
P	Polar coordinates
I	Incremental dimensions
Q	Q parameters
	Capture actual position
NO ENT	Skip dialog questions, delete words
ENT	Confirm entry and resume dialog
END □	End block
CE	Clear numerical entry or TNC error or delete error message
DEL □	Abort dialog, delete program section


HEIDENHAIN
01:03:24
ON
OFF
START SINGLE
START
RESET + START
PGM MGT
CALC
MOD
HELP
APPR DEP
FK
CHF
CR
RND
CT
CC
TOUCH PROBE
CYCL DEF
CYCL CALL
LBL SET
LBL CALL
STOP
TOOL DEF
TOOL CALL
PGM CALL
SHIFT
SPACE
RET
CE
DEL
NO ENT
ENT
END
GOTO
S %
F %

TNC models, software and features

This manual describes functions and features provided by the TNCs as of the following NC software numbers.

TNC model	NC software no.
TNC 426 CB, TNC 426 PB	280 476-xx
TNC 426 CF, TNC 426 PF	280 477-xx
TNC 426 M	280 476-xx
TNC 426 ME	280 477-xx
TNC 430 CA, TNC 430 PA	280 476-xx
TNC 430 CE, TNC 430 PE	280 477-xx
TNC 430 M	280 476-xx
TNC 430 ME	280 477-xx
TNC 410	286 060-xx
TNC 410	286 080-xx

The suffixes E and F indicate the export versions of the TNC The export versions of the TNC have the following limitations:

- Linear movement is possible in no more than 4 axes simultaneously.

The machine tool builder adapts the useable features of the TNC to his machine by setting machine parameters. Some of the functions described in this manual may not be among the features provided by your machine tool.

TNC functions that may not be available on your machine include:

- Probing function for the 3-D touch probe
- Digitizing option
- Tool measurement with the TT 130
- Rigid tapping
- Returning to the contour after an interruption

Please contact your machine tool builder to become familiar with the features of your machine.

Many machine manufacturers, as well as HEIDENHAIN, offer programming courses for the TNCs. We recommend these courses as an effective way of improving your programming skill and sharing information and ideas with other TNC users.

Touch Probe Cycles User's Manual:

All of the touch probe functions are described in a separate manual. Please contact HEIDENHAIN if you require a copy of this User's Manual. ID number: 329 203-xx.

Location of use

The TNC complies with the limits for a Class A device in accordance with the specifications in EN 55022, and is intended for use primarily in industrially-zoned areas.

New features of the NC software 280 476-xx

- Thread milling cycles 262 to 267 (see "Fundamentals of thread milling" on page 208)
- Tapping Cycle 209 with chip breaking (see "TAPPING WITH CHIP BREAKING (Cycle G209, not TNC 410)" on page 206)
- Cycle 247(see "DATUM SETTING (Cycle G247, not TNC 410)" on page 299)
- Entering two miscellaneous functions M (see "Entering Miscellaneous Functions M" on page 148)
- Program stop with M01 (see "Optional Program Run Interruption" on page 386)
- Starting NC programs automatically (see "Automatic Program Start (not TNC 410)" on page 383)
- Selecting the screen layout for pallet tables (see "Screen layout for executing pallet tables" on page 95)
- New columns in the tool table for managing TS calibration data (see "Entering tool data in tables" on page 101)
- Management of unlimited calibration data with the TS triggering touch probes (see User's Manual for Touch Probe Cycles)
- Cycles for automatic tool measurement with the TT tool touch probe in ISO (see User's Manual for Touch Probe Cycles)
- New Cycle 440 for measuring the axial displacement of a machine with the TT tool touch probe (see User's Manual for Touch Probe Cycles)
- Support of Teleservice functions (see "Teleservice (not TNC 410)" on page 418)
- Setting the display mode for blocks with more than one line, e.g. for cycle definitions (see "General User Parameters" on page 422)
- M142 (see "Erasing modal program information: M142 (not TNC 410)" on page 163)
- M143 (see "Erasing the basic rotation: M143 (not TNC 410)" on page 163)
- M144 (see "Compensating the machine's kinematic configuration for ACTUAL/NOMINAL positions at end of block: M144 (not TNC 410)" on page 171)
- External access with the LSV-2 interface (see "Permitting/ Restricting external access" on page 419)

Changed features of the NC software 280 476-xx

- The feed-rate unit for M136 was changed from µm/rev to mm/rev. (see “Feed rate in millimeters per spindle revolution: M136 (not TNC 410)” on page 159)
- The size of the contour memory for SL cycles was doubled. (see “SL Cycles Group II (not TNC 410)” on page 265)
- M91 and M92 are now also possible with tilted working plane. (see “Positioning in a tilted coordinate system” on page 306)
- Display of the NC program during the execution of pallet tables (see “Program Run, Full Sequence and Program Run, Single Block” on page 8) and (see “Screen layout for executing pallet tables” on page 95)

New/Changed Descriptions in this Manual

- TNCremoNT (see “Data transfer between the TNC and TNCremoNT” on page 398)
- Summary of input formats (see “Input format and unit of TNC functions” on page 443)
- Mid-program startup of pallet tables (see “Mid-program startup (block scan)” on page 380)
- Exchanging the buffer battery (see “Exchanging the Buffer Battery” on page 445)

Contents

13 Tables and Overviews 421

1

Introduction

1.1 The TNC 410, the TNC 426 and the TNC 430

HEIDENHAIN TNC controls are workshop-oriented contouring controls that enable you to program conventional machining operations right at the machine in an easy-to-use conversational programming language. They are designed for milling, drilling and boring machines, as well as for machining centers. The TNC 410 can control up to 4 axes, the TNC 426 up to 5 axes, and the TNC 430 up to 9 axes. You can also change the angular position of the spindle under program control.

An integrated hard disk provides storage for as many programs as you like, even if they were created off-line or by digitizing. For quick calculations you can call up the on-screen pocket calculator at any time.

Keyboard and screen layout are clearly arranged in such a way that the functions are fast and easy to use.

Programming: HEIDENHAIN conversational and ISO formats

HEIDENHAIN conversational programming is an especially easy method of writing programs. Interactive graphics illustrate the individual machining steps for programming the contour. If a production drawing is not dimensioned for NC, the HEIDENHAIN FK free contour programming does the necessary calculations automatically. Workpiece machining can be graphically simulated either during or before actual machining. It is also possible to program in ISO format or DNC mode.

You can also enter and test one program while the control is running another. With the TNC 426, TNC 430 it is also possible to test one program while another is being run.

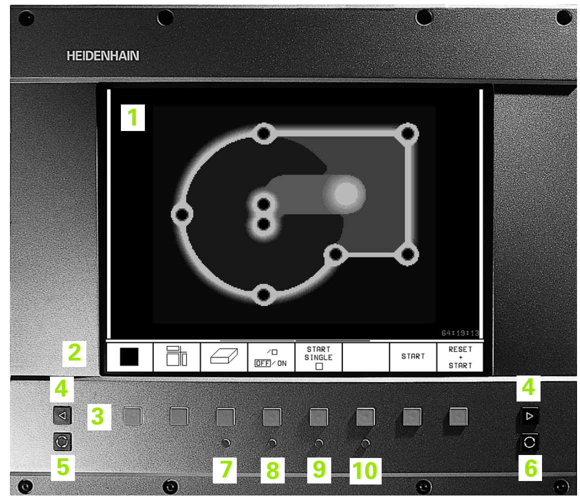
Compatibility

The TNC can execute all part programs that were written on HEIDENHAIN controls TNC 150 B and later.

1.2 Visual Display Unit and Keyboard

Visual display unit

The TNC is available with either a color CRT screen (BC 120) or a TFT flat panel display (BF 120). The figure at top right shows the keys and controls on the BC 120, and the figure at center right shows those of the BF 120.



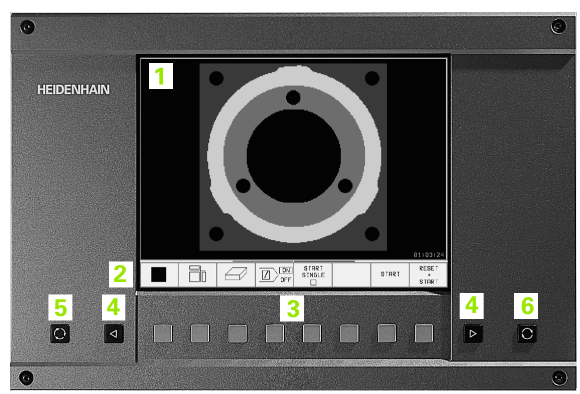
1 Header

   When the TNC is on, the selected operating modes are shown in the screen header: the machining mode at the left and the programming mode at right. The currently active mode is displayed in the larger box, where the dialog prompts and TNC messages also appear (unless the TNC is showing only graphics).

2 Soft keys

   In the footer the TNC indicates additional functions in a soft-key row. You can select these functions by pressing the keys immediately below them. The lines immediately above the soft-key row indicate the number of soft-key rows that can be called with the black arrow keys to the right and left. The line representing the active soft-key row is highlighted.

3 Soft key selector keys
4 Switching the soft-key rows
5 Setting the screen layout
6 Shift key for switchover between machining and programming modes



Keys on BC 120 only

7 Screen demagnetization; Exit main menu for screen settings
8 Select main menu for screen settings:
   - In the main menu: Move highlight downward
   - In the submenu: Reduce value or move picture to the left or downward
9
   - In the main menu: Move highlight upward
   - In the submenu: Increase value or move picture to the right or upward
10
   - In the main menu: Select submenu
   - In the submenu: Exit submenu

Main menu dialog	Function
BRIGHTNESS	Adjust brightness
CONTRAST	Adjust contrast
H-POSITION	Adjust horizontal position

Main menu dialog	Function
V-POSITION	Adjust vertical position
V-SIZE	Adjust picture height
SIDE-PIN	Correct barrel-shaped distortion
TRAPEZOID	Correct trapezoidal distortion
ROTATION	Correct tilting
COLOR TEMP	Adjust color temperature
R-GAIN	Adjust strength of red color
B-GAIN	Adjust strength of blue color
RECALL	No function

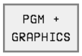
The BC 120 is sensitive to magnetic and electromagnetic noise, which can distort the position and geometry of the picture. Alternating fields can cause the picture to shift periodically or to become distorted.

Screen layout

You select the screen layout yourself: In the Programming and Editing mode of operation, for example, you can have the TNC show program blocks in the left window while the right window displays programming graphics (only TNC 410). The available screen windows depend on the selected operating mode.

To change the screen layout:

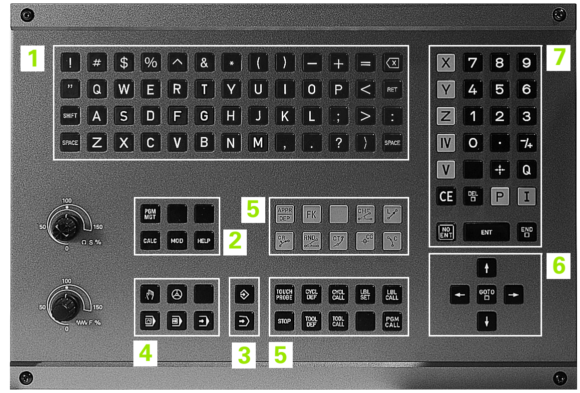
Press the SPLIT SCREEN key: The soft-key row shows the available layout options (see "Modes of Operation," page 6).



Select the desired screen layout.

Keyboard

The figure at right shows the keys of the keyboard grouped according to their functions:



1 Alphabetic keyboard for entering texts and file names, as well as for programming in ISO format
2
- File management
- Pocket calculator (not TNC 410)
- MOD functions
- HELP functions

3 Programming modes
4 Machine operating modes
5 Initiation of programming dialog
6 Arrow keys and GOTO jump command
7 Numerical input and axis selection

The functions of the individual keys are described on the inside front cover. Machine panel buttons, e.g. NC START, are described in the manual for your machine tool.

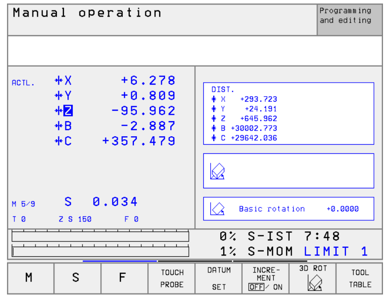
1.3 Modes of Operation

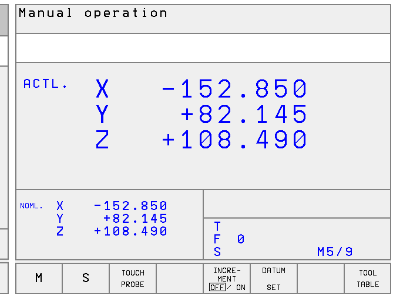
Manual Operation and Electronic Handwheel

The Manual Operation mode is required for setting up the machine tool. In this operating mode, you can position the machine axes manually or by increments, set the datums, and tilt the working plane.

The Electronic Handwheel mode of operation allows you to move the machine axes manually with the HR electronic handwheel.

Soft keys for selecting the screen layout (select as described above, TNC 410: see screen layout with program run, full sequence)



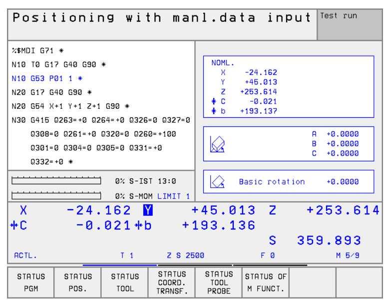


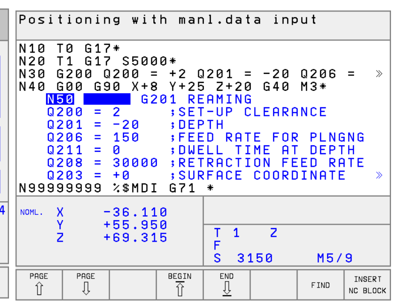
Screen windows	Soft key
Positions	POSITION
Left: positions, right: status display	POSITION + STATUS

Positioning with Manual Data Input (MDI)

This mode of operation is used for programming simple traversing movements, such as for face milling or pre-positioning. You can also define point tables for setting the digitizing range in this mode.

Soft keys for selecting the screen layout

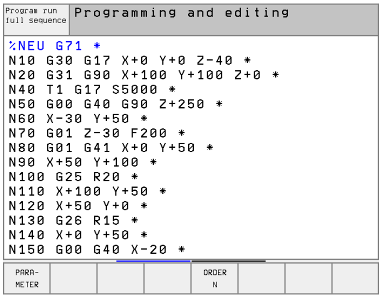


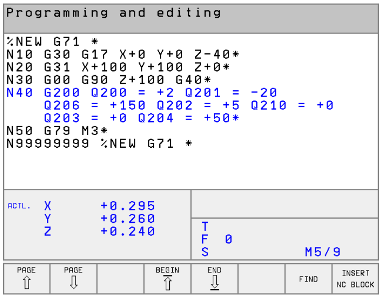


Screen windows	Soft key
Program	PGM
Left: program. Right: status display (only TNC 426, TNC 430)	PGM + STATUS
Left: program. Right: general program information (only TNC 410)	PGM + PGM STATUS
Left: program. Right: positions and coordinates (only TNC 410)	PGM + POS.DISP. STATUS
Left: program. Right: information on tools (only TNC 410)	PGM + TOOL STATUS
Left: program. Right: coordinate transformations (only TNC 410)	PGM + C.TRANS. STATUS

Programming and editing

In this mode of operation you can write your part programs. The various cycles and Q-parameter functions help you with programming and add necessary information.



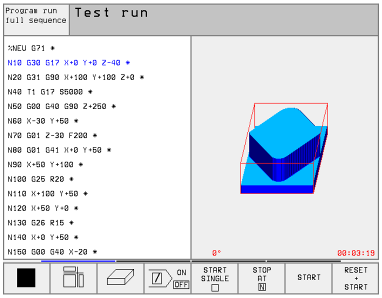


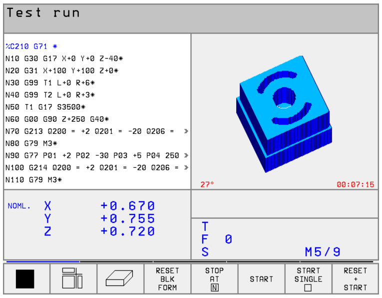
Soft keys for selecting the screen layout (only TNC 410)

Screen windows	Soft key
Program	PGM
Left: program. Right: help graphics for cycle programming	PGM + FIGURE
Left: program. Right: programming graphics	PGM + GRAPHICS
Interactive Programming graphics	GRAPHICS

Test Run

In the Test Run mode of operation, the TNC checks programs and program sections for errors, such as geometrical incompatibilities, missing or incorrect data within the program or violations of the work space. This simulation is supported graphically in different display modes.



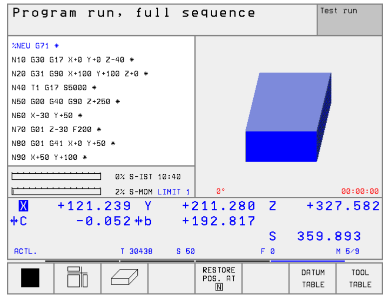


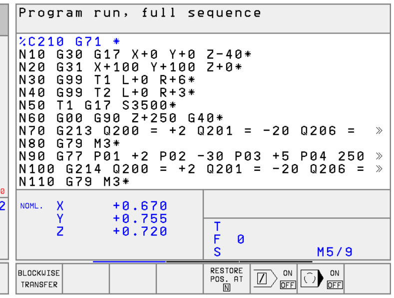
Soft keys for selecting the screen layout: see "Program Run, Full Sequence and Program Run, Single Block," page 8.

Program Run, Full Sequence and Program Run, Single Block

In the Program Run, Full Sequence mode of operation the TNC executes a part program continuously to its end or to a manual or programmed stop. You can resume program run after an interruption.

In the Program Run, Single Block mode of operation you execute each block separately by pressing the machine START button.





Soft keys for selecting the screen layout

Screen windows	Soft key
Program	PGM
Left: program. Right: status display (only TNC 426, TNC 430)	PGM + STATUS
Left: program. Right: graphics (only TNC 426, TNC 430)	PGM + GRAPHICS
Graphics (only TNC 426, TNC 430)	GRAPHICS
Left: program. Right: general program information (only TNC 410)	PGM + PGM STATUS
Left: program. Right: positions and coordinates (only TNC 410)	PGM + POS.DISP. STATUS
Left: program. Right: information on tools (only TNC 410)	PGM + TOOL STATUS
Left: program. Right: coordinate transformations (only TNC 410)	PGM + C.TRANS. STATUS
Left: program. Right: tool measurement (only TNC 410)	PGM + T.PROBE STATUS

Soft keys for selecting the screen layout for pallet tables (only TNC 426, TNC 430): see next page.

Soft keys for selecting the screen layout for pallet tables (only TNC 426, TNC 430)

Screen windows	Soft key
Pallet table	PALLET
Left: program. Right: pallet table	PGM + PALLET
Left: pallet table. Right: status	PALLET + STATUS
Left: pallet table. Right: graphics	PALLET + GRAPHICS

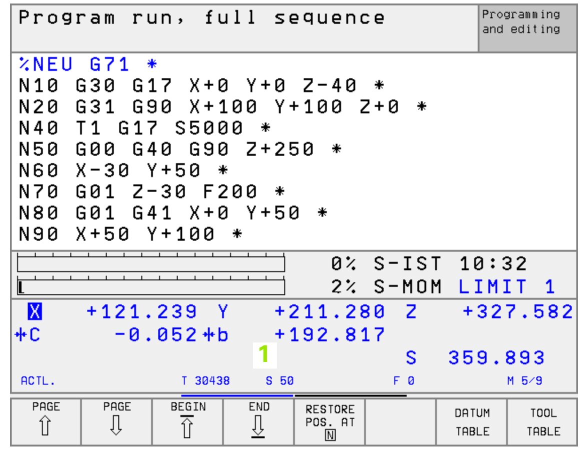
1.4 Status Displays

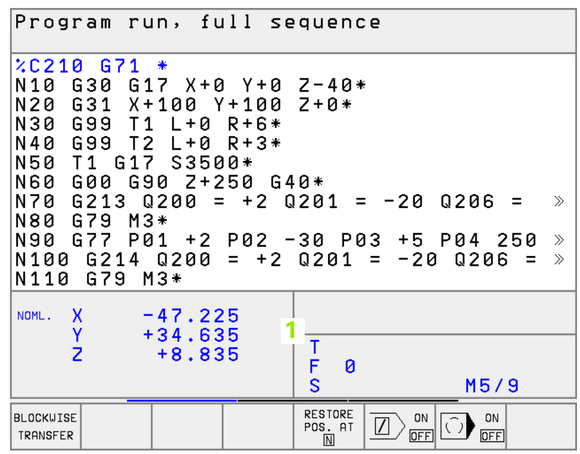
“General” status display

The status display **1** informs you of the current state of the machine tool. It is displayed automatically in the following modes of operation:

- Program Run, Single Block and Program Run, Full Sequence, except if the screen layout is set to display graphics only, and
- Positioning with Manual Data Input (MDI).

In the Manual mode and Electronic Handwheel mode the status display appears in the large window.





Information in the status display

Symbol	Meaning
ACTL.	Actual or nominal coordinates of the current position
X Y Z	Machine axes; the TNC displays auxiliary axes in lower-case letters. The sequence and quantity of displayed axes is determined by the machine tool builder. Refer to your machine manual for more information
F S **M**	The displayed feed rate in inches corresponds to one tenth of the effective value. Spindle speed S, feed rate F and active M functions
*	Program run started
	Axis locked
	Axis can be moved with the handwheel
	Axes are moving in a tilted working plane (only TNC 426, TNC 430)
	Axes are moving under a basic rotation

Additional status displays

The additional status displays contain detailed information on the program run. They can be called in all operating modes except for the Programming and Editing mode of operation.

To switch on the additional status display:

Call the soft-key row for screen layout.

PGM + STATUS

Select the layout option for the additional status display.

To select an additional status display:

Shift the soft-key rows until the STATUS soft keys appear.

STATUS PGM

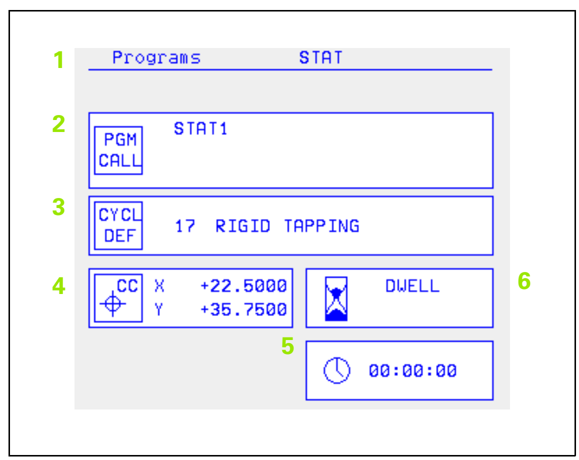
Select the desired additional status display, e.g. general program information.

You can choose between several additional status displays with the following soft keys:

STATUS PGM

General program information

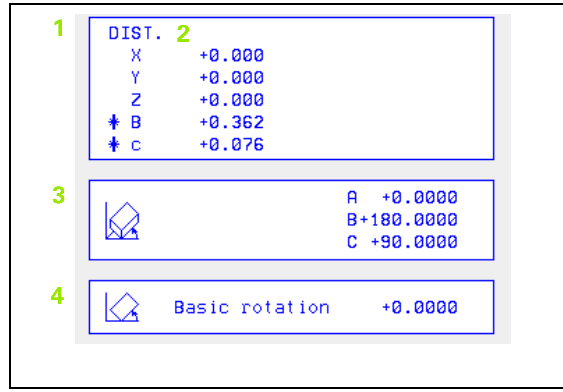
1 Name of main program
2 Active programs
3 Active machining cycle
4 Circle center CC (pole)
5 Operating time
6 Dwell time counter



STATUS POS.

Positions and coordinates

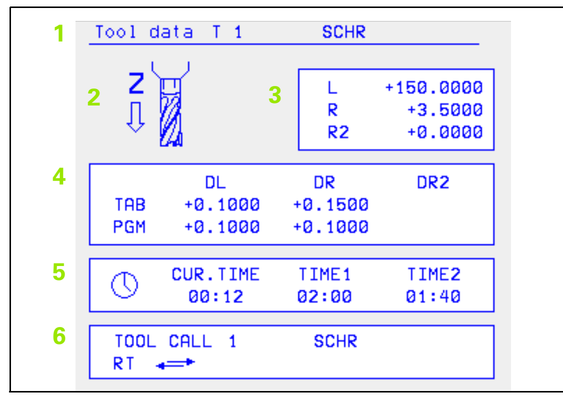
1 Position display
2 Type of position display, e.g. actual position
3 Tilting angle for the working plane (only TNC 426, TNC 430)
4 Angle of a basic rotation



STATUS TOOL

Information on tools

1 ■ T: Tool number and name
  ■ RT: Number and name of a replacement tool
2 Tool axis
3 Tool length and radii
4 Oversizes (delta values) from TOOL CALL (PGM) and the tool table (TAB)
5 Tool life, maximum tool life (TIME 1) and maximum tool life for TOOL CALL (TIME 2)
6 Display of the active tool and the (next) replacement tool

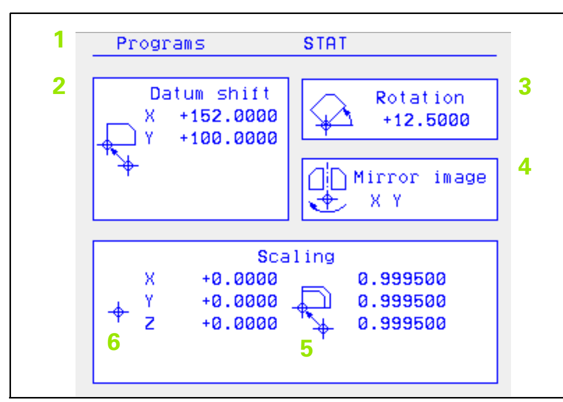


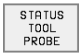
STATUS COORD. TRANSF.

Coordinate transformations

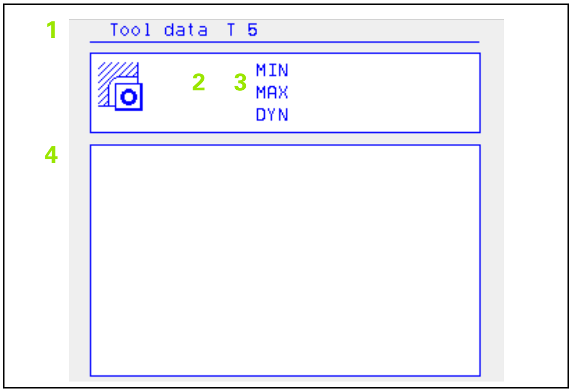
1 Name of main program
2 Active datum shift (Cycle 7)
3 Active rotation angle (Cycle 10)
4 Mirrored axes (Cycle 8)
5 Active scaling factor(s) (Cycles 11 / 26)
6 Scaling datum

(see "Coordinate Transformation Cycles" on page 294)

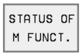




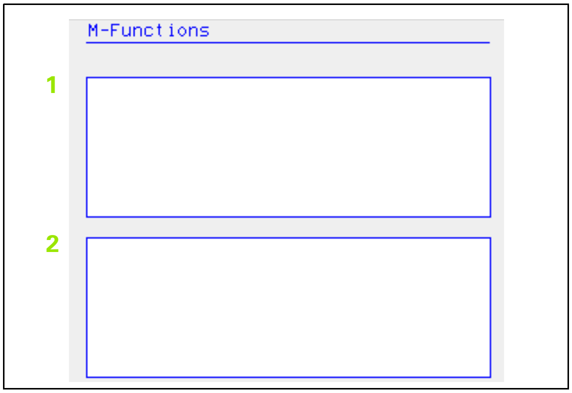
Tool measurement



1 Number of the tool to be measured
2 Display whether the tool radius or the tool length is being measured
3 MIN and MAX values of the individual cutting edges and the result of measuring the rotating tool (DYN = dynamic measurement)
4 Cutting edge number with the corresponding measured value. If the measured value is followed by an asterisk, the allowable tolerance in the tool table was exceeded



Active miscellaneous functions M (not TNC 410)



1 List of the active M functions with fixed meaning.
2 List of the active M functions with function assigned by machine manufacturer.

1.5 Accessories: HEIDENHAIN 3-D Touch Probes and Electronic Handwheels

3-D touch probes

With the various HEIDENHAIN 3-D touch probe systems you can:

- Automatically align workpieces
- Quickly and precisely set datums
- Measure the workpiece during program run
- Digitize 3-D surfaces (option), and
- Measure and inspect tools

All of the touch probe functions are described in a separate manual. Please contact HEIDENHAIN if you require a copy of this User's Manual. ID number: 329 203-xx.

TS 220, TS 630 and TS 632 touch trigger probes

These touch probes are particularly effective for automatic workpiece alignment, datum setting, workpiece measurement and for digitizing. The TS 220 transmits the triggering signals to the TNC via cable and is a cost-effective alternative for applications where digitizing is not frequently required.

The TS 630 and TS 632 feature infrared transmission of the triggering signal to the TNC. This makes them highly convenient for use on machines with automatic tool changers.

Principle of operation: HEIDENHAIN triggering touch probes feature a wear-resistant optical switch that generates an electrical signal as soon as the stylus is deflected. This signal is transmitted to the TNC, which stores the current position of the stylus as an actual value.

During digitizing the TNC generates a program containing straight line blocks in HEIDENHAIN format from a series of measured position data. You can then output the program to a PC for further processing with the SUSA evaluation software. This evaluation software enables you to calculate male/female transformations or correct the program to account for special tool shapes and radii that differ from the shape of the stylus tip. If the tool has the same radius as the stylus tip you can run these programs immediately.

TT 130 tool touch probe for tool measurement

The TT 130 is a triggering 3-D touch probe for tool measurement and inspection. Your TNC provides three cycles for this touch probe with which you can measure the tool length and radius automatically either with the spindle rotating or stopped. The TT 130 features a particularly rugged design and a high degree of protection, which make it insensitive to coolants and swarf. The triggering signal is generated by a wear-resistant and highly reliable optical switch.

HR electronic handwheels

Electronic handwheels facilitate moving the axis slides precisely by hand. A wide range of traverses per handwheel revolution is available. Apart from the HR 130 and HR 150 integral handwheels, HEIDENHAIN also offers the HR 410 portable handwheel (see figure at center right).

2

Manual Operation and Setup

2.1 Switch-on, Switch-Off

Switch-on

Switch-on and Traversing the Reference Points can vary depending on the individual machine tool. Refer to your machine manual.

Switch on the power supply for control and machine. The TNC automatically initiates the following dialog

Memory Test

The TNC memory is automatically checked.

Power Interrupted

CE — TNC message that the power was interrupted—clear the message.

Translate PLC program

The PLC program of the TNC is automatically compiled.

Relay Ext. DC Voltage Missing

I — Switch on external dc voltage. The TNC checks the functioning of the EMERGENCY STOP circuit.

Manual Operation
Traverse Reference Points

I — Cross the reference points manually in the displayed sequence: For each axis press the machine START button, or

X Y — Cross the reference points in any sequence: Press and hold the machine axis direction button for each axis until the reference point has been traversed, or

Z Y — Cross the reference points with several axes at the same time: Use soft keys to select the axes (axes are then shown highlighted on the screen), and then press the machine START button (only TNC 410).

The TNC is now ready for operation in the Manual Operation mode.

Additional functions for the TNC 426, TNC 430

The reference points need only be traversed if the machine axes are to be moved. If you intend only to write, edit or test programs, you can select the Programming and Editing or Test Run modes of operation immediately after switching on the control voltage.

You can then traverse the reference points later by pressing the PASS OVER REFERENCE soft key in the Manual Operation mode.

Traversing the reference point in a tilted working plane

The reference point of a tilted coordinate system can be traversed by pressing the machine axis direction buttons. The "tilting the working plane" function must be active in the Manual Operation mode, see "To activate manual tilting:," page 29. The TNC then interpolates the corresponding axes.

The NC START button is not effective. Pressing this button may result in an error message.

Make sure that the angle values entered in the menu for tilting the working plane match the actual angles of the tilted axis.

Switch-off

To prevent data being lost at switch-off, you need to run down the operating system as follows:

- Select the Manual mode.

- Select the function for shutting down, confirm again with the YES soft key.
- When the TNC displays the message **`Now you can switch off the TNC`** in a superimposed window, you may cut off the power supply to the TNC.

Inappropriate switch-off of the TNC can lead to data loss.

2.2 Moving the Machine Axes

Note

Traversing with the machine axis direction buttons is a machine-dependent function. The machine tool manual provides further information.

To traverse with the machine axis direction buttons:

Select the Manual Operation mode.

X

Press the machine axis-direction button and hold it as long as you wish the axis to move, or

X

and

I

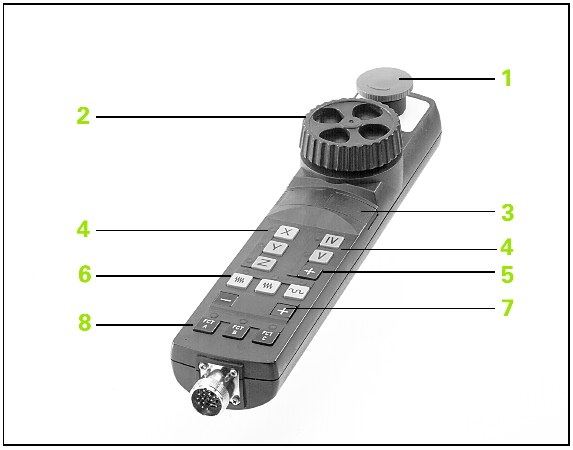
Move the axis continuously: Press and hold the machine axis direction button, then press the machine START button

0

To stop the axis, press the machine STOP button.

You can move several axes at a time with these two methods. You can change the feed rate at which the axes are traversed with the F soft key, see "Spindle Speed S, Feed Rate F and Miscellaneous Functions M," page 23.

Traversing with the HR 410 electronic handwheel



The portable HR 410 handwheel is equipped with two permissive buttons. The permissive buttons are located below the star grip.

You can only move the machine axes when a permissive button is depressed (machine-dependent function).

The HR 410 handwheel features the following operating elements:

1 EMERGENCY STOP
2 Handwheel
3 Permissive buttons
4 Axis address keys
5 Actual-position-capture key
6 Keys for defining the feed rate (slow, medium, fast; the feed rates are set by the machine tool builder)
7 Direction in which the TNC moves the selected axis
8 Machine function (set by the machine tool builder)

The red indicators show the axis and feed rate you have selected.

It is also possible to move the machine axes with the handwheel during a program run.

To move an axis:

Select the Electronic Handwheel operating mode.

Press and hold the permissive button.

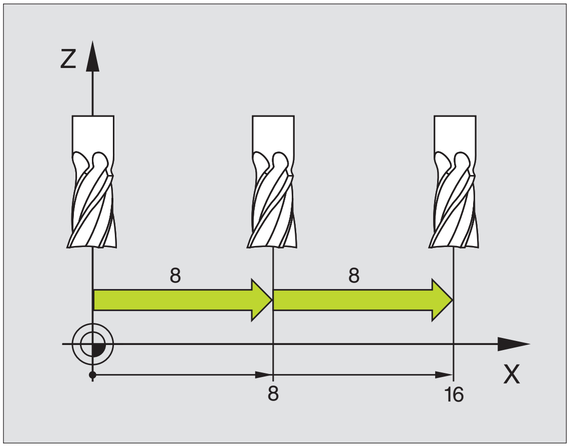
X Select the axis.

Select the feed rate.

+ or − Move the active axis in the positive or negative direction.

Incremental jog positioning

With incremental jog positioning you can move a machine axis by a preset distance.



Select the Manual or Electronic Handwheel mode of operation.

INCRE-
MENT
OFF / ON

Select incremental jog positioning: Switch the INCREMENT soft key to ON

Jog increment =

8 ENT

Enter the jog increment in millimeters, i.e. 8 mm.

X

Press the machine axis direction button as often as desired.

2.3 Spindle Speed S, Feed Rate F and Miscellaneous Functions M

Function

In the operating modes Manual Operation and Electronic Handwheel, you can enter the spindle speed S, feed rate F and the miscellaneous functions M with soft keys. The miscellaneous functions are described in Chapter 7 "Programming: Miscellaneous Functions."

> The machine tool builder determines which miscellaneous functions M are available on your TNC and what effects they have.

Entering values

Spindle speed S, miscellaneous function M

S	To enter the spindle speed, press the S soft key.

```
Spindle speed S =
```

1000 I	Enter the desired spindle speed and confirm your entry with the machine START button.

The spindle speed S with the entered rpm is started with a miscellaneous function M. Proceed in the same way to enter a miscellaneous function M.

Feed rate F

After entering a feed rate F, you must confirm your entry with the ENT key instead of the machine START button.

The following is valid for feed rate F:

- If you enter F=0, then the lowest feed rate from MP1020 is effective
- F is not lost during a power interruption

Changing the spindle speed and feed rate

With the override knobs you can vary the spindle speed S and feed rate F from 0% to 150% of the set value.

> The override dial for spindle speed is only functional on machines with infinitely variable spindle drive.

2.4 Datum Setting (Without a 3-D Touch Probe)

Note

For datum setting with a 3-D touch probe, refer to the new Touch Probe Cycles Manual.

You fix a datum by setting the TNC position display to the coordinates of a known position on the workpiece.

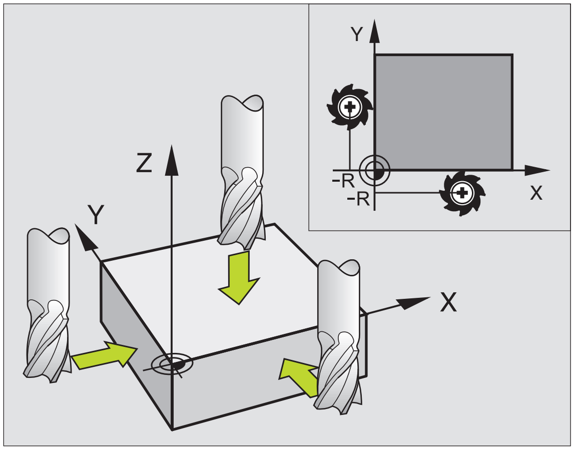
Preparation

- Clamp and align the workpiece.
- Insert the zero tool with known radius into the spindle.
- Ensure that the TNC is showing actual position values.

Datum setting

Fragile workpiece?

If the workpiece surface must not be scratched, you can lay a metal shim of known thickness d on it. Then enter a tool axis datum value that is larger than the desired datum by the value d.



Select the **Manual Operation** mode.

X Y Z Move the tool slowly until it touches the workpiece surface.

Select an axis (all axes can also be selected via the ASCII keyboard)

```
Datum Set Z=
```

0 ENT Zero tool in spindle axis: Set the display to a known workpiece position (here, 0) or enter the thickness *d* of the shim. In the tool axis, offset the tool radius.

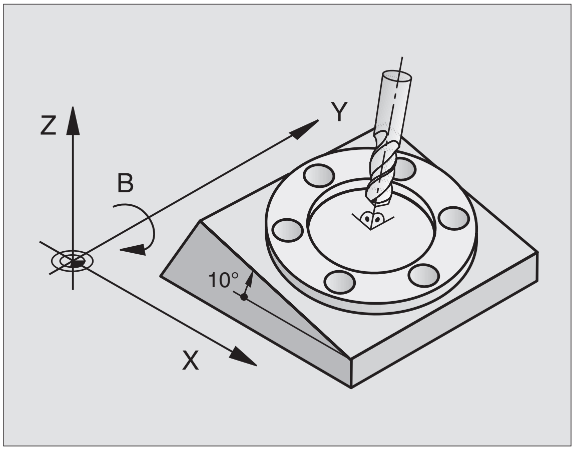
Repeat the process for the remaining axes.

If you are using a preset tool, set the display of the tool axis to the length *L* of the tool or enter the sum *Z*=*L*+*d*.

2.5 Tilting the Working Plane (not TNC 410)

Application, function

The functions for tilting the working plane are interfaced to the TNC and the machine tool by the machine tool builder. With some swivel heads and tilting tables, the machine tool builder determines whether the entered angles are interpreted as coordinates of the tilt axes or as angular components of a tilted plane. Refer to your machine manual.



The TNC supports the tilting functions on machine tools with swivel heads and/or tilting tables. Typical applications are, for example, oblique holes or contours in an oblique plane. The working plane is always tilted around the active datum. The program is written as usual in a main plane, such as the X/Y plane, but is executed in a plane that is tilted relative to the main plane.

There are two functions available for tilting the working plane:

- 3-D ROT soft key in the Manual mode and Electronic Handwheel mode, see “To activate manual tilting:,” page 29
- Tilting under program control, Cycle **G80 WORKING PLANE** in the part program (see “WORKING PLANE (Cycle G80, not TNC 410)” on page 304)

The TNC functions for “tilting the working plane” are coordinate transformations in which the working plane is always perpendicular to the direction of the tool axis.

When tilting the working plane, the TNC differentiates between two machine types:

- **Machines with tilting tables:**
  - You must tilt the workpiece into the desired position for machining by positioning the tilting table, for example with a G0 block.
  - The position of the transformed tool axis **does not change** in relation to the machine-based coordinate system. Thus if you rotate the table—and therefore the workpiece—by 90° for example, the coordinate system **does not rotate.** If you press the Z+ axis direction button in the Manual Operation mode, the tool moves in Z+ direction.
  - In calculating the transformed coordinate system, the TNC considers only the mechanically influenced offsets of the particular tilting table (the so-called “translational” components).

- **Machines with swivel heads**
  - You must bring the tool into the desired position for machining by positioning the swivel head, for example with a G0 block.
  - The position of the transformed tool axis changes in relation to the machine-based coordinate system. Thus if you rotate the swivel head of your machine—and therefore the tool—in the B axis by 90° for example, the coordinate system rotates also. If you press the Z+ axis direction button in the Manual Operation mode, the tool moves in X+ direction of the machine-based coordinate system.
  - In calculating the transformed coordinate system, the TNC considers both the mechanically influenced offsets of the particular swivel head (the so-called “translational” components) and offsets caused by tilting of the tool (3-D tool length compensation).

Traversing the reference points in tilted axes

With tilted axes, you use the machine axis direction buttons to cross over the reference points. The TNC interpolates the corresponding axes. Be sure that the function for tilting the working plane is active in the Manual Operation mode and the actual angle of the tilted axis was entered in the menu field.

Setting the datum in a tilted coordinate system

After you have positioned the rotary axes, set the datum in the same way as for a non-tilted system. The TNC then converts the datum for the tilted coordinate system. If your machine tool features axis control, the angular values for this calculation are taken from the actual position of the rotary axis.

You must not set the datum in the tilted working plane if in machine parameter 7500 bit 3 is set. If you do, the TNC will calculate the wrong offset.

If your machine tool is not equipped with axis control, you must enter the actual position of the rotary axis in the menu for manual tilting: The actual positions of one or several rotary axes must match the entry. Otherwise the TNC will calculate an incorrect datum.

Datum setting on machines with rotary tables

The behavior of the TNC during datum setting depends on the machine. Refer to your machine manual.

The TNC automatically shifts the datum if you rotate the table and the tilted working plane function is active:

- **MP 7500, bit 3=0**
  To calculate the datum, the TNC uses the difference between the REF coordinate during datum setting and the REF coordinate of the tilting axis after tilting. The method of calculation is to be used when you have clamped your workpiece in proper alignment when the rotary table is in the 0° position (REF value).
- **MP 7500, bit 3=1**
  If you rotate the table to align a workpiece that has been clamped in an unaligned position, the TNC must no longer calculate the offset of the datum from the difference of the REF coordinates. Instead of the difference from the 0° position, the TNC uses the REF value of the tilting table after tilting. In other words, it assumes that you have properly aligned the workpiece before tilting.

MP 7500 is effective in the machine parameter list, or, if available, in the descriptive tables for tilted axis geometry. Refer to your machine manual.

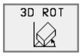
Position display in a tilted system

The positions displayed in the status window (**ACTL.** and **NOML.**) are referenced to the tilted coordinate system.

Limitations on working with the tilting function

- The touch probe function Basic Rotation cannot be used.
- PLC positioning (determined by the machine tool builder) is not possible.
- Positioning blocks with M91/M92 are not permitted.

To activate manual tilting:

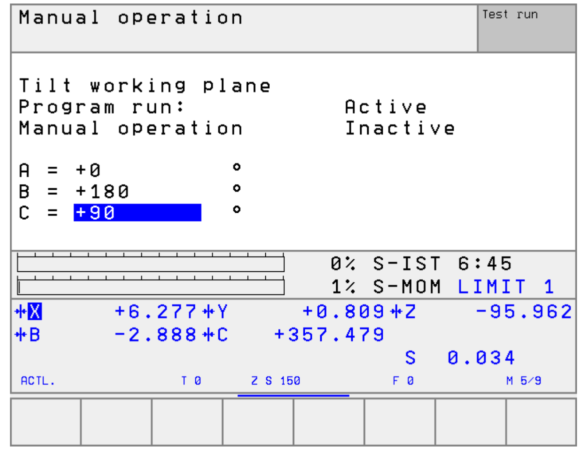


To select manual tilting, press the 3-D ROT soft key. You can now select the desired menu items with the arrow keys

Enter the tilt angle.

To set the desired operating mode in menu option "Tilt working plane" to Active, select the menu option and shift with the ENT key.

To conclude entry, press the END key.



To reset the tilting function, set the desired operating modes in menu "Tilt working plane" to Inactive.

If the tilted working plane function is active and the TNC moves the machine axes in accordance with the tilted axes, the status display shows the symbol

If you set the function "Tilt working plane" for the operating mode Program Run to Active, the tilt angle entered in the menu becomes active in the first block of the part program. If you are using Cycle 19 **WORKING PLANE** in the part program, the angular values defined in the cycle (starting at the cycle definition) are effective. Angular values entered in the menu will be overwritten.

3

Positioning with Manual Data Input (MDI)

3.1 Programming and Executing Simple Machining Operations

The Positioning with Manual Data Input mode of operation is particularly convenient for simple machining operations or pre-positioning of the tool. It enables you to write a short program in HEIDENHAIN conversational programming or in ISO format, and execute it immediately. You can also call TNC cycles. The program is stored in the file $MDI. In the operating mode Positioning with MDI, the additional status displays can also be activated.

Positioning with Manual Data Input (MDI)

Select the Positioning with MDI mode of operation. Program the file $MDI as you wish.

To start program run, press the machine START button.

Limitations for TNC 410

The following functions are not available:

- Tool radius compensation
- Programming and program run graphics
- Programmable probe functions
- Subprograms, program section repeats
- Contouring functions **G06**, **G02** and **G03** with R, **G24** and **G25**
- Program call with %

Limitations for TNC 426, TNC 430

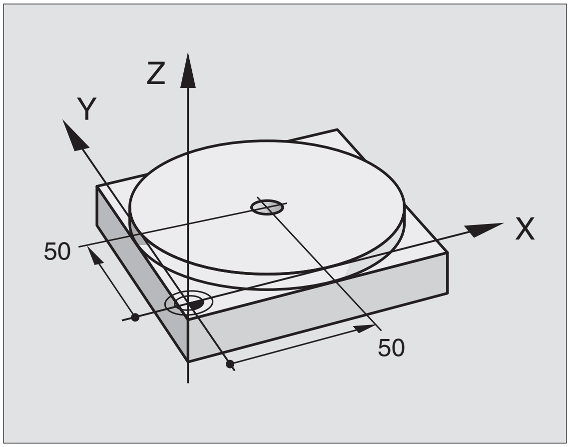
The following functions are not available:

- Program call with %
- Program run graphics

Example 1

A hole with a depth of 20 mm is to be drilled into a single workpiece. After clamping and aligning the workpiece and setting the datum, you can program and execute the drilling operation in a few lines.

First you pre-position the tool with straight-line blocks to the hole center coordinates at a setup clearance of 5 mm above the workpiece surface. Then drill the hole with Cycle G83 Pecking.



`%$MDI G71 *`	
`N10 G99 T1 L+0 R+5 *`	Define tool: zero tool, radius 5
`N20 T1 G17 S2000 *`	Call tool: tool axis Z
	Spindle speed 2000 rpm
`N30 G00 G40 G90 Z+200 *`	Retract tool (rapid traverse)
`N40 X+50 Y+50 M3 *`	Move the tool at rapid traverse to a position above the hole
	Spindle on
`N50 G01 Z+2 F2000 *`	Position tool to 2 mm above hole
`N60 G83`	Define Cycle G83 PECKING:
`P01 +2`	Set-up clearance of the tool above the hole
`P02 -20`	Total hole depth (Algebraic sign=working direction)
`P03 +10`	Depth of each infeed before retraction
`P04 0.5`	Dwell time in seconds at the hole bottom
`P05 250 *`	Feed rate for pecking
`N70 G79 *`	Call Cycle G83 PECKING
`N80 G00 G40 Z+200 M2 *`	Retract the tool
`N99999 %$MDI G71 *`	End of program

For details on the straight-line function G00 (see “Straight line at rapid traverse G00 Straight line with feed rate G01 F. . .” on page 127), for Cycle G83 PECKING (see “PECKING (Cycle G83)” on page 185).

Example 2: Correcting workpiece misalignment on machines with rotary tables

Use the 3-D touch probe to rotate the coordinate system. See "Touch Probe Cycles in the Manual and Electronic Handwheel Operating Modes," section "Compensating workpiece misalignment," in the new Touch Probes Cycles User's Manual.

Write down the rotation angle and cancel the Basic Rotation.

Select operating mode: Positioning with MDI.

Select the axis of the rotary table, enter the rotation angle you wrote down previously and set the feed rate. For example: **`G00 G40 G90 C+2.561 F50`**

Conclude entry.

Press the machine START button: The rotation of the table corrects the misalignment.

Protecting and erasing programs in $MDI

The $MDI file is generally intended for short programs that are only needed temporarily. Nevertheless, you can store a program, if necessary, by proceeding as described below:

Select the Programming and Editing mode of operation.

PGM MGT — To call the file manager, press the PGM MGT key (program management).

Move the highlight to the $MDI file.

COPY ABC → XYZ — To select the file copying function, press the COPY soft key.

Target file =

BOREHOLE — Enter the name under which you want to save the current contents of the $MDI file.

ENT — TNC 410: Start copying by pressing the ENT key

EXECUTE — TNC 426 B, TNC430: Press the EXECUTE soft key to start copying

END — To close the file manager, press the END soft key.

Erasing the contents of the $MDI file is done in a similar way: Instead of copying the contents, however, you erase them with the DELETE soft key. The next time you select the operating mode Positioning with MDI, the TNC will display an empty $MDI file.

> **TNC 426, TNC 430:** If you wish to delete $MDI, then
> - you must not have selected the Positioning with MDI mode (not even in the background).
> - you must not have selected the $MDI file in the Programming and Editing mode.

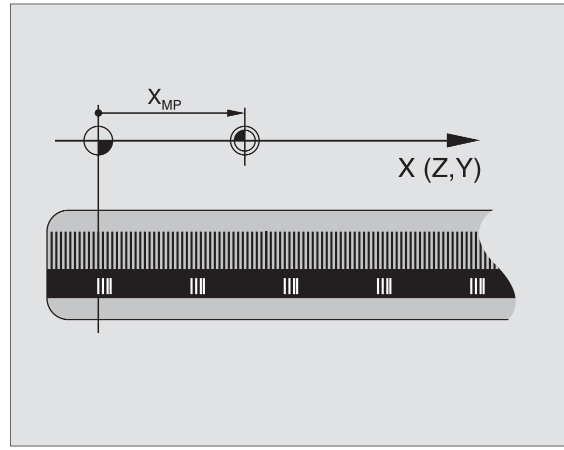
For further information, see "Copying a single file," page 58.

4

Programming: Fundamentals of NC, File Management, Programming Aids, Pallet Management

4.1 Fundamentals

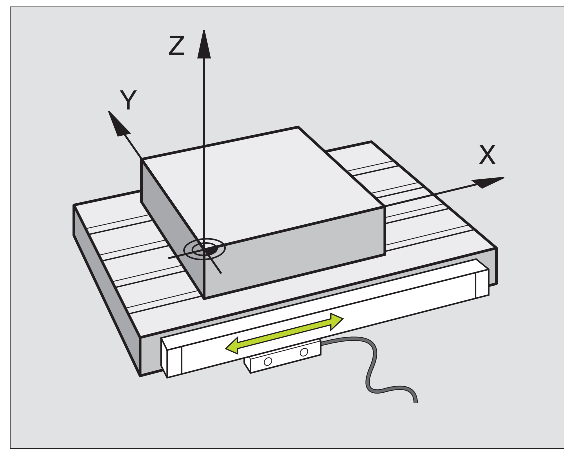
Position encoders and reference marks



The machine axes are equipped with position encoders that register the positions of the machine table or tool. When a machine axis moves, the corresponding position encoder generates an electrical signal. The TNC evaluates this signal and calculates the precise actual position of the machine axis.

If there is a power interruption, the calculated position will no longer correspond to the actual position of the machine slide. The control can re-establish this relationship with the aid of reference marks when power is returned. The scales of the position encoders contain one or more reference marks that transmit a signal to the TNC when the axes pass over them. From the signal the TNC identifies that position as the machine-axis reference point and can re-establish the assignment of displayed positions to machine axis positions.

Linear encoders are generally used for linear axes. Rotary tables and tilt axes have angle encoders. If the position encoders feature distance-coded reference marks, you only need to move each axis a maximum of 20 mm (0.8 in.) for linear encoders, and 20° for angle encoders, to re-establish the assignment of the displayed positions to machine axis positions.

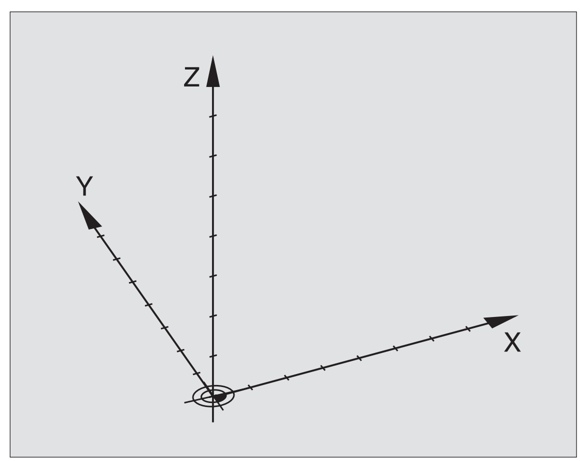


Reference system

A reference system is required to define positions in a plane or in space. The position data are always referenced to a predetermined point and are described through coordinates.

The Cartesian coordinate system (a rectangular coordinate system) is based on the three coordinate axes X, Y and Z. The axes are mutually perpendicular and intersect at one point called the datum. A coordinate identifies the distance from the datum in one of these directions. A position in a plane is thus described through two coordinates, and a position in space through three coordinates.

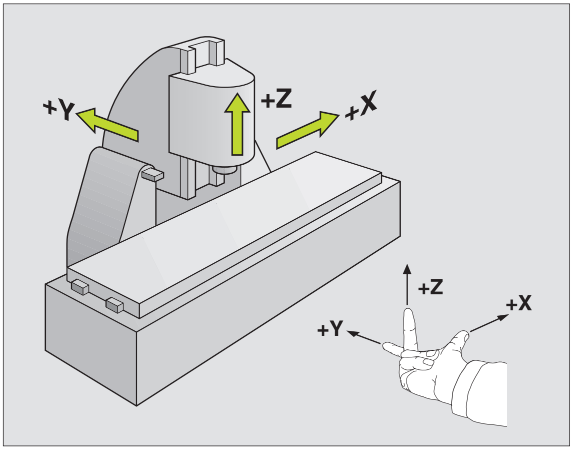
Coordinates that are referenced to the datum are referred to as absolute coordinates. Relative coordinates are referenced to any other known position (datum) you define within the coordinate system. Relative coordinate values are also referred to as incremental coordinate values.

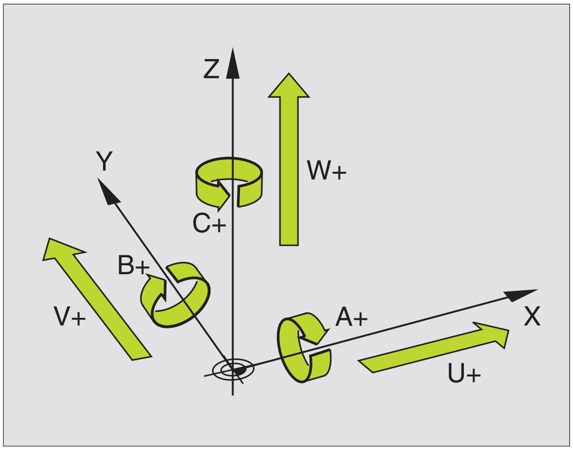


Reference system on milling machines

When using a milling machine, you orient tool movements to the Cartesian coordinate system. The illustration at right shows how the Cartesian coordinate system describes the machine axes. The figure at center right illustrates the “right-hand rule” for remembering the three axis directions: the middle finger is pointing in the positive direction of the tool axis from the workpiece toward the tool (the Z axis), the thumb is pointing in the positive X direction, and the index finger in the positive Y direction.

The TNC 410 can control a maximum of 4 axes, the TNC 426 a maximum of 5 axes and the TNC 430 a maximum of 9 axes. The axes U, V and W are secondary linear axes parallel to the main axes X, Y and Z, respectively. Rotary axes are designated as A, B and C. The illustration at lower right shows the assignment of secondary axes and rotary axes to the main axes.





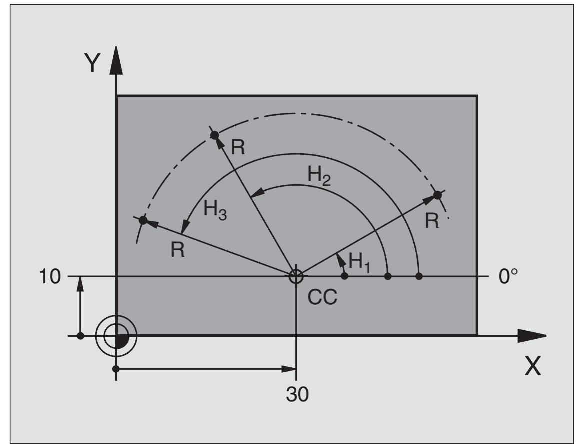
Polar coordinates

If the production drawing is dimensioned in Cartesian coordinates, you also write the part program using Cartesian coordinates. For parts containing circular arcs or angles it is often simpler to give the dimensions in polar coordinates.

While the Cartesian coordinates X, Y and Z are three-dimensional and can describe points in space, polar coordinates are two-dimensional and describe points in a plane. Polar coordinates have their datum at the pole. A position in a plane can be clearly defined by the

- Polar Radius, the distance from the pole to the position, and the
- Polar Angle, the size of the angle between the reference axis and the line that connects the pole with the position.

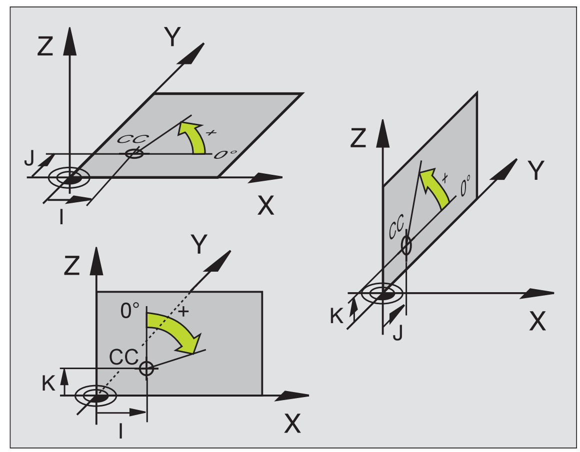
See figure at upper right.



Definition of pole and angle reference axis

The pole is set by entering two Cartesian coordinates in one of the three planes. These coordinates also set the reference axis for the polar angle H.

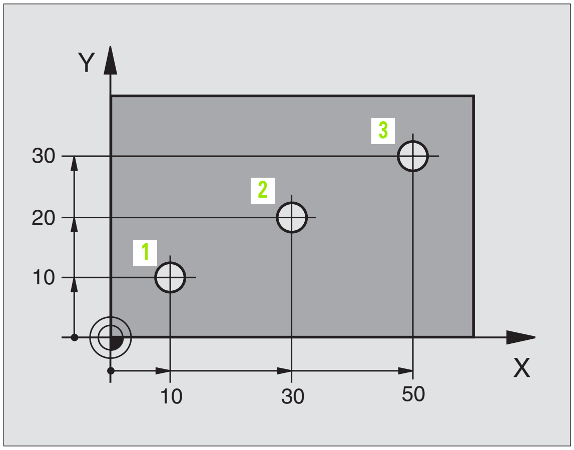
Coordinates of the pole (plane)	Reference axis of the angle
I and J	+X
J and K	+Y
K and I	+Z



Absolute and incremental workpiece positions

Absolute workpiece positions

Absolute coordinates are position coordinates that are referenced to the datum of the coordinate system (origin). Each position on the workpiece is uniquely defined by its absolute coordinates.



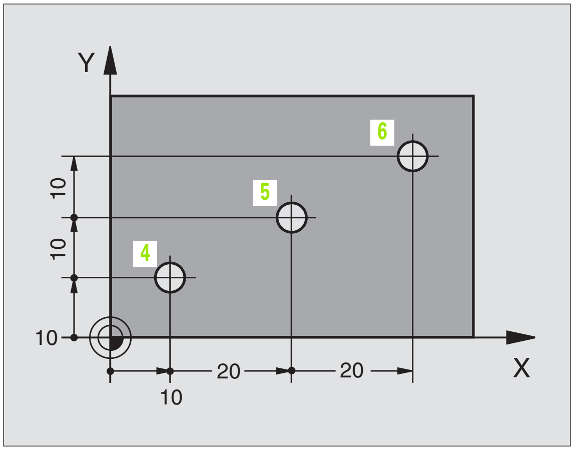
Example 1: Holes dimensioned in absolute coordinates

Hole **1**	Hole **2**	Hole **3**
X = 10 mm	X = 30 mm	X = 50 mm
Y = 10 mm	Y = 20 mm	Y = 30 mm

Incremental workpiece positions

Incremental coordinates are referenced to the last programmed nominal position of the tool, which serves as the relative (imaginary) datum. When you write a part program in incremental coordinates, you thus program the tool to move by the distance between the previous and the subsequent nominal positions. Incremental coordinates are therefore also referred to as chain dimensions.

To program a position in incremental coordinates, enter the function G91 before the axis.



Example 2: Holes dimensioned in incremental coordinates

Absolute coordinates of hole **4**

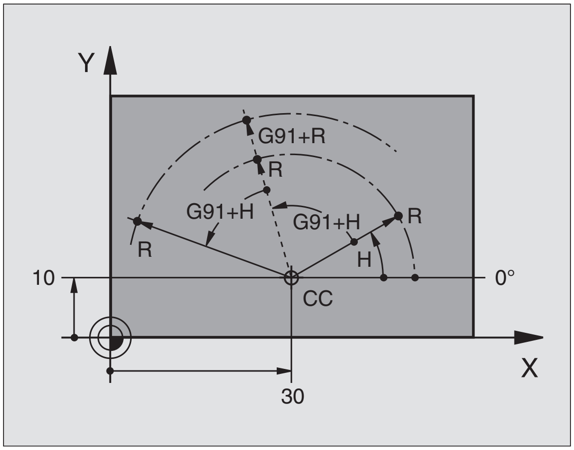
X = 10 mm
Y = 10 mm

Hole **5**, referenced to **4**	Hole **6**, referenced to **5**
G91 X= 20 mm	G91 X= 20 mm
G91 Y= 10 mm	G91 Y= 10 mm

Absolute and incremental polar coordinates

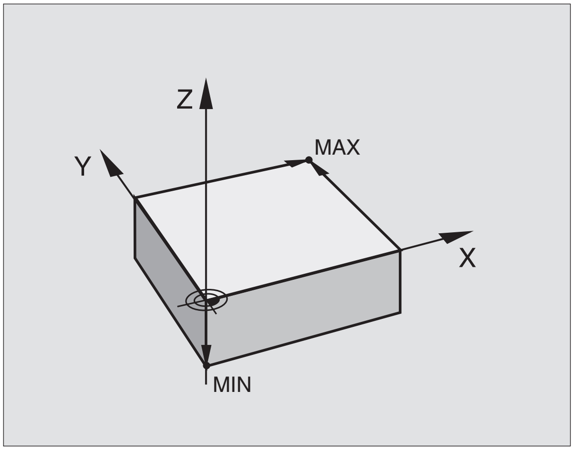
Absolute polar coordinates always refer to the pole and the reference axis.

Incremental polar coordinates always refer to the last programmed nominal position of the tool.



Setting the datum

A production drawing identifies a certain form element of the workpiece, usually a corner, as the absolute datum. Before setting the datum, you align the workpiece with the machine axes and move the tool in each axis to a known position relative to the workpiece. You then set the TNC display either to zero or to a predetermined position value. This establishes the reference system for the workpiece, which will be used for the TNC display and your part program.



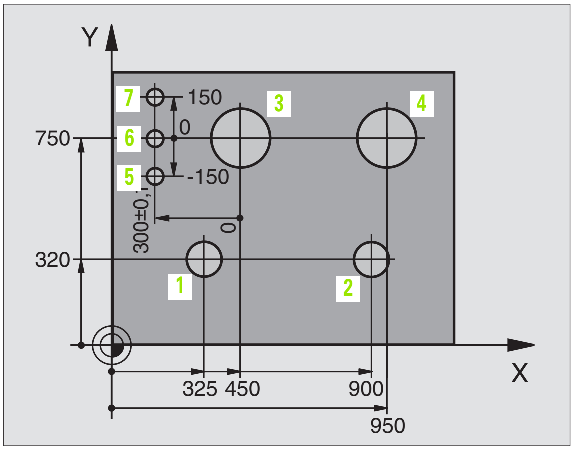
If the production drawing is dimensioned in relative coordinates, simply use the coordinate transformation cycles. (see "Coordinate Transformation Cycles" on page 294).

If the production drawing is not dimensioned for NC, set the datum at a position or corner on the workpiece which is suitable for deducing the dimensions of the remaining workpiece positions.

The fastest, easiest and most accurate way of setting the datum is by using a 3-D touch probe from HEIDENHAIN. See the new Touch Probe Cycles User's Manual, chapter "Setting the Datum with a 3-D Touch Probe."

Example

The workpiece drawing at right shows holes (**1** to **4**) whose dimensions are shown with respect to an absolute datum with the coordinates X=0 Y=0. The holes (**5** to **7**) are dimensioned with respect to a relative datum with the absolute coordinates X=450, Y=750. With the **DATUM SHIFT** cycle you can temporarily set the datum to the position X=450, Y=750, to be able to program the holes (**5** to **7**) without further calculations.



4.2 File Management: Fundamentals

Files

Files in the TNC	Type
Programs	
In HEIDENHAIN format	.H
In ISO format	.I
Tables for	
Tools	.T
Tool changers	.TCH
Pallets (not TNC 410)	.P
Datums	.D
Points	.PNT
Cutting data (not TNC 410)	.CDT
Cutting materials, workpiece materials (not TNC 410)	.TAB
Texts as	
ASCII files (not TNC 410)	.A

When you write a part program on the TNC, you must first enter a file name. The TNC saves the program as a file with the same name. The TNC can also save texts and tables as files.

The TNC provides a special file management window in which you can easily find and manage your files. Here you can call, copy, rename and erase files.

In the TNC 410 you can manage a max. 64 files with a total of up to 256 KB.

The TNC 426, TNC 430 can manage any number of files. However, their total size must not exceed **1500 MB**.

File names

When you store programs, tables and texts as files, the TNC adds an extension to the file name, separated by a period. This extension indicates the file type.

PROG20	.H
File name	File type
Maximum Length	See table "Files in the TNC."

Data backup TNC 426, TNC 430

We recommend saving newly written programs and files on a PC at regular intervals.

You can do this with the free backup program TNCBACK.EXE from HEIDENHAIN. Your machine tool builder can provide you with a copy of TNCBACK.EXE.

In addition, you need a floppy disk on which all machine-specific data, such as PLC program, machine parameters, etc., are stored. Please contact your machine tool builder for more information on both the backup program and the floppy disk.

Saving the contents of the entire hard disk (up to 1500 MB) can take up to several hours. In this case, it is a good idea to save the data outside of working hours, (e.g. overnight), or to use the PARALLEL EXECUTE function to copy in the background while you work.

4.3 Standard File Management TNC 426, TNC 430

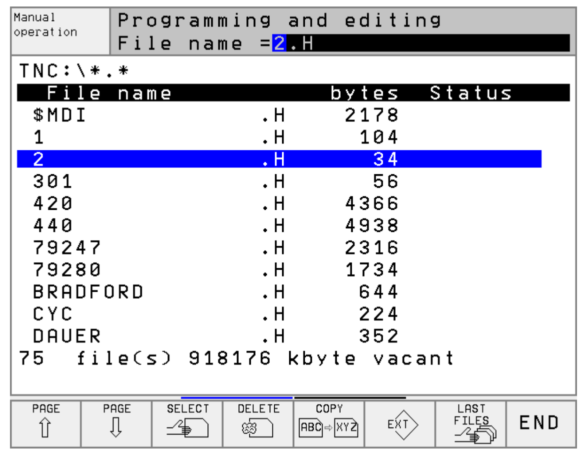
Note

> The standard file management is best if you wish to save all files in one directory, or if you are well practiced in the file management of old TNC controls.
>
> To use the standard file management, set the MOD function **PGM MGT** (see "Configuring PGM MGT (not TNC 410)" on page 406) to **Standard**.

Calling the file manager

PGM MGT

Press the PGM MGT key: The TNC displays the file management window (see figure at right)



The window shows you all of the files that are stored in the TNC. Each file is shown with additional information:

Display	Meaning
FILE NAME	Name with up to 16 characters and file type
BYTE	File size in bytes
STATUS	File properties:
E	Program is selected in the Programming and Editing mode of operation.
S	Program is selected in the Test Run mode of operation.
M	Program is selected in a program run operating mode.
P	File is protected against editing and erasure.

Selecting a file

PGM MGT — Call the file manager.

Use the arrow keys or the arrow soft keys to move the highlight to the file you wish to select:

Moves the highlight up or down **file by file** in the window.

PAGE ⇩ PAGE ⇧ — Moves the highlight up or down **page by page** in the window.

SELECT or ENT — To select the file: Press the SELECT soft key or the ENT key.

Deleting a file

PGM MGT — Call the file manager.

Use the arrow keys or the arrow soft keys to move the highlight to the file you wish to delete:

Moves the highlight up or down **file by file** in the window.

PAGE ⇩ PAGE ⇧ — Moves the highlight up or down **page by page** in the window.

DELETE — To delete the file: Press the DELETE soft key.

Delete ..... file?

YES — Confirm with the YES soft key.

NO — Abort with the NO soft key.

Copying a file

PGM MGT — Call the file manager.

Use the arrow keys or the arrow soft keys to move the highlight to the file you wish to copy:

Moves the highlight up or down **file by file** in the window.

PAGE ⇧ PAGE ⇩ — Moves the highlight up or down **page by page** in the window.

COPY ABC → XYZ — To copy the file: Press the COPY soft key.

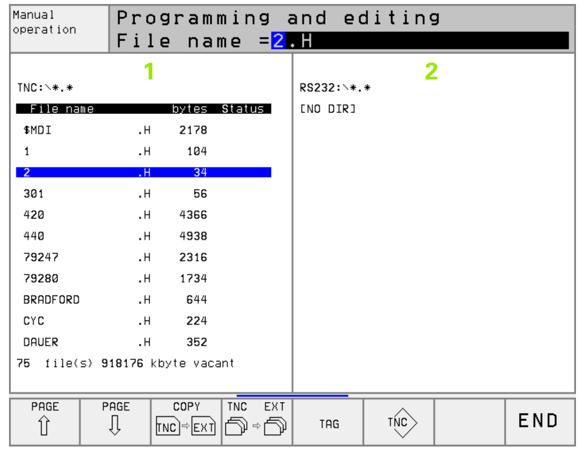
Target file =

Enter the new name, and confirm your entry with the EXECUTE soft key or the ENT key. A status window appears on the TNC, informing about the copying progress. As long as the TNC is copying, you can no longer work, or

If you wish to copy very long programs, enter the new file name and confirm with the PARALLEL EXECUTE soft key. The file will now be copied in the background, so you can continue to work while the TNC is copying.

Data transfer to or from an external data medium

Before you can transfer data to an external data medium, you must setup the data interface (see “Setting the Data Interfaces for TNC 426, TNC 430” on page 395).



PGM MGT — Call the file manager.

EXT — Activate data transfer: Press the EXT soft key. In the left half of the screen **(1)** the TNC shows all files saved on its hard disk. In the right half of the screen **(2)** it shows all files saved on the external data medium.

Use the arrow keys to highlight the file(s) that you want to transfer:

↑ ↓ Moves the highlight up and down within a window.

← → Moves the highlight from the left to the right window, and vice versa.

If you wish to copy from the TNC to the external data medium, move the highlight in the left window to the file to be transferred.

If you wish to copy from the external data medium to the TNC, move the highlight in the right window to the file to be transferred.

Tagging functions	Soft key
Tag a single file	TAG FILE
Tag all files	TAG ALL FILES
Untag a single file	UNTAG FILE
Untag all files	UNTAG ALL FILES
Copy all tagged files	COPY TAG

COPY ABC ⇒ XYZ — Transfer a single file: Press the COPY soft key, or

TAG — Transfer several files: Press the TAG soft key, or

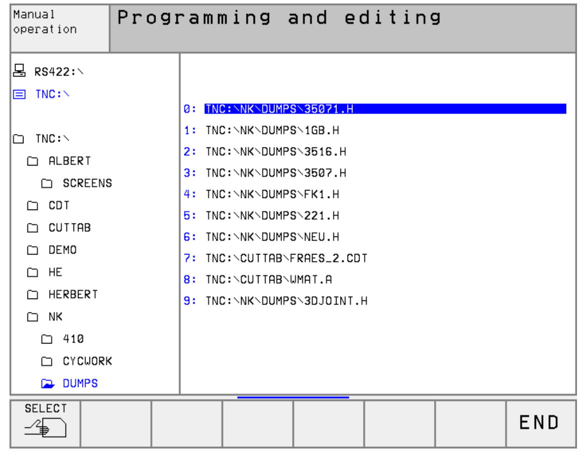
KOPIEREN TNC ⇒ EXT — Transfer all files: Press the TNC => EXT soft key.

Confirm with the EXECUTE soft key or with the ENT key. A status window appears on the TNC, informing about the copying progress, or

If you wish to transfer more than one file or longer files, press the PARALLEL EXECUTE soft key. The TNC then copies the file in the background.

TNC — To stop transfer, press the TNC soft key. The standard file manager window is displayed again.

Selecting one of the last 10 files selected



PGM MGT — Call the file manager.

LAST FILES — Display the last 10 files selected: Press the LAST FILES soft key.

Use the arrow keys to move the highlight to the file you wish to select:

↑ ↓ — Move the highlight up or down.

SELECT or ENT — To select the file: Press the SELECT soft key or the ENT key.

Renaming a file

PGM MGT — Call the file manager.

Use the arrow keys or the arrow soft keys to move the highlight to the file you wish to rename:

↑ ↓ — Moves the highlight up or down **file by file** in the window.

PAGE ⇧ PAGE ⇩ — Moves the highlight up or down **page by page** in the window.

RENAME ABC = XYZ — Press the RENAME soft key to select the renaming function

`Target file =`

Enter the name of the new file and confirm your entry with the ENT key or EXECUTE soft key.

Converting an FK program into HEIDENHAIN conversational format

PGM MGT

Call the file manager.

Use the arrow keys or the arrow soft keys to move the highlight to the file you wish to convert:

Moves the highlight up or down **file by file** in the window.

PAGE ⇧ PAGE ⇩

Moves the highlight up or down **page by page** in the window.

CONVERT FK->H

Convert the file: Press the CONVERT FK –> H soft key.

Target file =

Enter the name of the new file and confirm your entry with the ENT key or EXECUTE soft key.

Protecting a file / Canceling file protection

PGM MGT — Call the file manager.

Use the arrow keys or arrow soft keys to move the highlight to the file you wish to protect or whose protection you wish to cancel:

Moves the highlight up or down **file by file** in the window.

PAGE PAGE — Moves the highlight up or down **page by page** in the window.

PROTECT — To enable file protection: Press the PROTECT soft key. The file now has status P, or

UNPROTECT — Press the UNPROTECT soft key to cancel file protection. The P status is canceled.

4.4 Expanded File Management TNC 426, TNC 430

Note

Use the advanced file manager if you wish to keep your files in individual directories.

To use it, set the MOD function PGM MGT (see "Configuring PGM MGT (not TNC 410)" on page 406).

See also "File Management: Fundamentals" on page 43.

Directories

To ensure that you can easily find your files, we recommend that you organize your hard disk into directories. You can divide a directory into further directories, which are called subdirectories.

The TNC can manage up to 6 directory levels!

If you save more than 512 files in one directory, the TNC no longer sorts them alphabetically!

Directory names

The name of a directory can contain up to 8 characters and does not have an extension. If you enter more than 8 characters for the directory name, the TNC will display an error message.

Paths

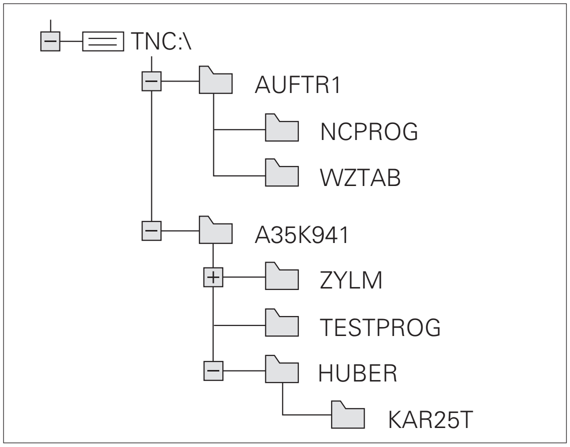
A path indicates the drive and all directories and subdirectories under which a file is saved. The individual names are separated by a backslash "\".

Example

On drive **TNC:** the subdirectory AUFTR1 was created. Then, in the directory **AUFTR1** the directory NCPROG was created and the part program PROG1.I was copied into it. The part program now has the following path:

TNC:\AUFTR1\NCPROG\PROG1.I

The chart at right illustrates an example of a directory display with different paths.



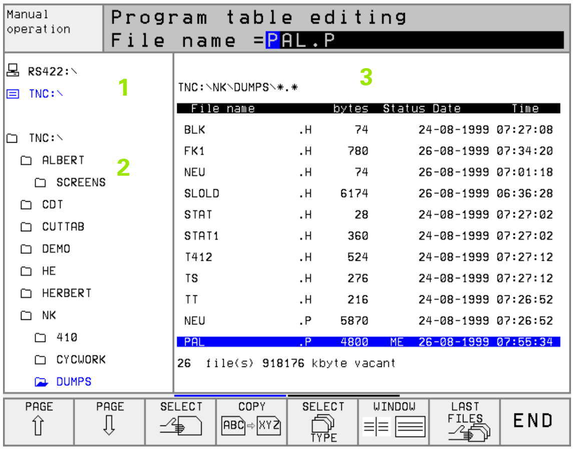
Overview: Functions of the expanded file manager

Function	Soft key
Copy (and convert) individual files	COPY ABC XYZ
Display a specific file type	SELECT TYPE
Display the last 10 files that were selected	LAST FILES
Erase a file or directory	DELETE
Tag a file	TAG
Renaming a file	RENAME ABC = XYZ
Protect a file against editing and erasure	PROTECT
Cancel file protection	UNPROTECT
Network drive management (Ethernet option only)	NET
Copy a directory	COPY DIR
Display all the directories of a particular drive	SHOW TREE
Delete directory with all its subdirectories	DELETE ALL

Calling the file manager

PGM MGT

Press the PGM MGT soft key: The TNC displays the file management window. (The figure at top right shows the basic settings. If the TNC shows a different screen layout, press the WINDOW soft key.)



The narrow window at left shows three drives **(1)**. If the TNC is connected to a network, it also displayed the connected network drives. Drives designate devices with which data are stored or transferred. One drive is the hard disk of the TNC. Other drives are the interfaces (RS232, RS422, Ethernet), which can be used, for example, to connect a personal computer. The selected (active) drive is shown in a different color.

In the lower part of the narrow window the TNC shows all directories **(2)** of the selected drive. A drive is always identified by a file symbol to the left and the directory name to the right. The control displays a subdirectory to the right of and below its parent directory. The selected (active) directory is depicted in a different color.

The wide window at right **3** shows you all of the files that are stored in the selected directory. Information for each file is displayed in a table to the right.

Display	Meaning
FILE NAME	Name with up to 16 characters and file type
BYTE	File size in bytes
STATUS	File properties:
E	Program is selected in the Programming and Editing mode of operation.
S	Program is selected in the Test Run mode of operation.
M	Program is selected in a program run operating mode.
P	File is protected against editing and erasure.
DATE	Date the file was last changed
TIME	Time the file was last changed

Selecting drives, directories and files

PGM MGT	Call the file manager.

With the arrow keys or the soft keys, you can move the highlight to the desired position on the screen:

← →	Moves the highlight from the left to the right window, and vice versa.
↓ ↑	Moves the highlight up and down within a window.
PAGE ⇩ PAGE ⇧	Moves the highlight one page up or down within a window.

1st step: Select a drive

Move the highlight to the desired drive in the left window:

SELECT or ENT	Select a drive: Press the SELECT soft key or the ENT key.

2nd step: Select a directory

Move the highlight to the desired directory in the left-hand window — the right-hand window automatically shows all files stored in the highlighted directory.

3rd step: select a file

SELECT TYPE

Press the SELECT TYPE soft key.

SHOW .H

Press the soft key for the desired file type, or

SHOW ALL

Press the SHOW ALL soft key to display all files, or

4*.H ENT

Use wild card characters, e.g. to show all files of the file type .H that begin with 4.

Move the highlight to the desired file in the right window

SELECT

or

ENT

The selected file is opened in the operating mode from which you have called the file manager: Press the SELECT soft key or the ENT key.

Creating a new directory (only possible on the drive TNC:\)

Move the highlight in the left window to the directory in which you want to create a subdirectory.

NEW ENT

Enter the new file name, and confirm with ENT.

Create \NEW directory?

YES

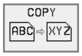
Press the YES soft key to confirm, or

NO

Abort with the NO soft key.

Copying a single file

▶ Move the highlight to the file you wish to copy.



▶ Press the COPY soft key to select the copying function.

▶ Enter the name of the destination file and confirm your entry with the ENT key or EXECUTE soft key: The TNC copies the file into the active directory. The original file is retained, or

▶ Press the PARALLEL EXECUTE soft key to copy the file in the background. Copying in the background permits you to continue working while the TNC is copying. This can be useful if you are copying very large files that take a long time. While the TNC is copying in the background you can press the INFO PARALLEL EXECUTE soft key (under MORE FUNCTIONS, second soft-key row) to check the progress of copying.

Copying a table

If you are copying tables, you can overwrite individual lines or columns in the target table with the REPLACE FIELDS soft key. Prerequisites:

- The target table must exist.
- The file to be copied must only contain the columns or lines you want to replace.

> The **REPLACE FIELDS** soft key does not appear when you want to overwrite the table in the TNC with an external data transfer software, such as TNCremoNT. Copy the externally created file into a different directory, and then copy the desired fields with the TNC file management.

Example

With a tool presetter you have measured the length and radius of 10 new tools. The tool presetter then generates the tool table TOOL.T with 10 lines (for the 10 tools) and the columns

- Tool number (column **T**)
- Tool length (column **L**)
- Tool radius (column **R**).

Copy this file to a directory other than the one containing the previous TOOL.T. If you wish to copy this file over the existing table using the TNC file management, the TNC asks if you wish to overwrite the existing TOOL.T tool table:

▶ If you press the YES soft key, the TNC will completely overwrite the current TOOL.T tool table. After this copying process the new TOOL.T table consists of 10 lines. The only remaining columns in the table are tool number, tool length and tool radius.

▶ Or, if you press the REPLACE FIELDS soft key, the TNC merely overwrites the first 10 lines of the columns number, length and radius in the TOOL.T file. The TNC does not change the data in the other lines and columns.

Copying a directory

Move the highlight in the left window onto the directory you want to copy. Instead of the COPY soft key, press the COPY DIR soft key. Subdirectories are also copied at the same time.

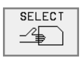
Choosing one of the last 10 files selected.

Call the file manager.

Display the last 10 files selected: Press the LAST FILES soft key.

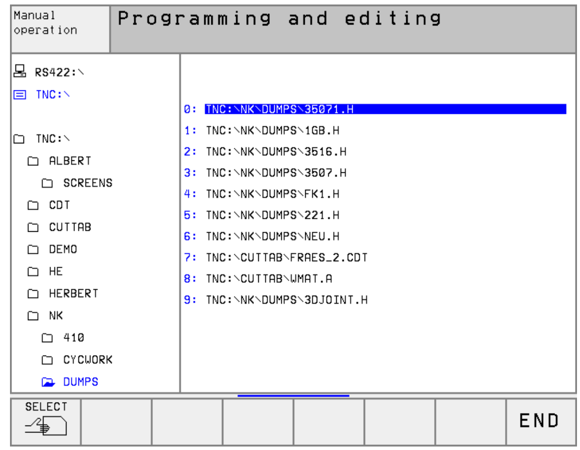
Use the arrow keys to move the highlight to the file you wish to select:

Moves the highlight up and down within a window.



or

Select a drive: Press the SELECT soft key or the ENT key.



Deleting a file

- Move the highlight to the file you want to delete.

- To select the erasing function, press the DELETE soft key. The TNC inquires whether you really intend to erase the file.
- To confirm, press the YES soft key;
- To abort erasure, press the NO soft key.

Deleting a directory

- Delete all files and subdirectories stored in the directory that you wish to erase.
- Move the highlight to the directory you want to delete.

DELETE

- To select the erasing function, press the DELETE soft key. The TNC inquires whether you really intend to erase the directory.
- To confirm, press the YES soft key;
- To abort erasure, press the NO soft key.

Tagging files

Tagging functions	Soft key
Tag a single file	TAG FILE
Tag all files in the directory	TAG ALL FILES
Untag a single file	UNTAG FILE
Untag all files	UNTAG ALL FILES
Copy all tagged files	COPY TAG

Some functions, such as copying or erasing files, can not only be used for individual files, but also for several files at once. To tag several files, proceed as follows:

Move the highlight to the first file.

TAG

To display the tagging functions, press the TAG soft key.

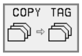
TAG FILE

Tag a file by pressing the TAG FILE soft key.

Move the highlight to the next file you wish to tag:

TAG FILE

You can tag several files in this way, as desired.

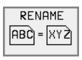


To copy the tagged files, press the COPY TAG soft key, or

Delete the tagged files by pressing END to end the marking function, and then the DELETE soft key to delete the tagged files.

Renaming a file

▶ Move the highlight to the file you want to rename.



▶ Select the renaming function.

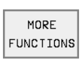
▶ Enter the new file name; the file type cannot be changed.

▶ To execute renaming, press the ENT key.

Additional functions

Protecting a file / Canceling file protection

▶ Move the highlight to the file you want to protect.



▶ To select the additional functions, press the MORE FUNCTIONS soft key.

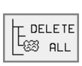
▶ To enable file protection, press the PROTECT soft key. The file now has status P.

▶ To cancel file protection, proceed in the same way using the UNPROTECT soft key.

Erase a directory together with all its subdirectories and files.

▶ Move the highlight in the left window onto the directory you want to erase.

▶ To select the additional functions, press the MORE FUNCTIONS soft key.

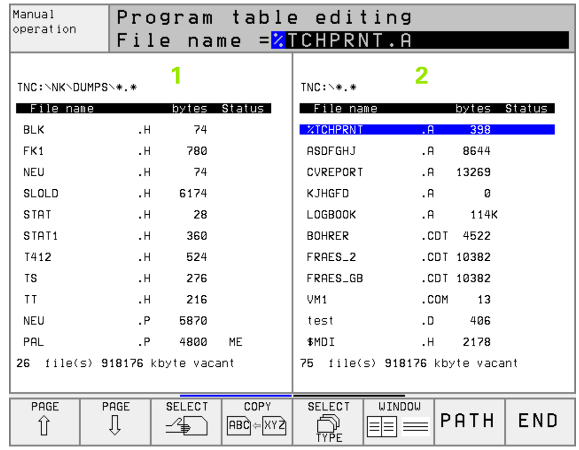


▶ Press DELETE ALL to erase the directory together with its subdirectories.

▶ To confirm, press the YES soft key; To abort erasure, press the NO soft key.

Data transfer to or from an external data medium

Before you can transfer data to an external data medium, you must setup the data interface (see “Setting the Data Interfaces for TNC 426, TNC 430” on page 395).



PGM MGT — Call the file manager.

WINDOW — Select the screen layout for data transfer: press the WINDOW soft key. In the left half of the screen **(1)** the TNC shows all files saved on its hard disk. In the right half of the screen **(2)** it shows all files saved on the external data medium.

Use the arrow keys to highlight the file(s) that you want to transfer:

↑ ↓ — Moves the highlight up and down within a window.

← → — Moves the highlight from the left to the right window, and vice versa.

If you wish to copy from the TNC to the external data medium, move the highlight in the left window to the file to be transferred.

If you wish to copy from the external data medium to the TNC, move the highlight in the right window to the file to be transferred.

COPY ABC ⇒ XYZ — Transfer a single file: Press the COPY soft key, or

TAG — Transfer several files: Press the TAG soft key (in the second soft-key row, see “Tagging files,” page 60), or

KOPIEREN TNC ⇒ EXT — Transfer all files: Press the TNC => EXT soft key.

Confirm with the EXECUTE soft key or with the ENT key. A status window appears on the TNC, informing about the copying progress, or

If you wish to transfer more than one file or longer files, press the PARALLEL EXECUTE soft key. The TNC then copies the file in the background.

To end data transfer, move the highlight into left window and then press the WINDOW soft key. The standard file manager window is displayed again.

To select another directory, press the PATH soft key and then select the desired directory using the arrow keys and the ENTkey!

Copying files into another directory

- Select the screen layout with the two equally sized windows.
- To display directories in both windows, press the PATH soft key.

In the right window

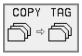
- Move the highlight to the directory into which you wish to copy the files, and display the files in this directory with the ENT key.

In the left window

- Select the directory with the files that you wish to copy and press ENT to display them.

- Display the file tagging functions.

- Move the highlight to the file you want to copy and tag it. You can tag several files in this way, as desired.



- Copy the tagged files into the target directory.

Additional tagging functions: see “Tagging files,” page 60.

If you have marked files in the left and right windows, the TNC copies from the directory in which the highlight is located.

Overwriting files

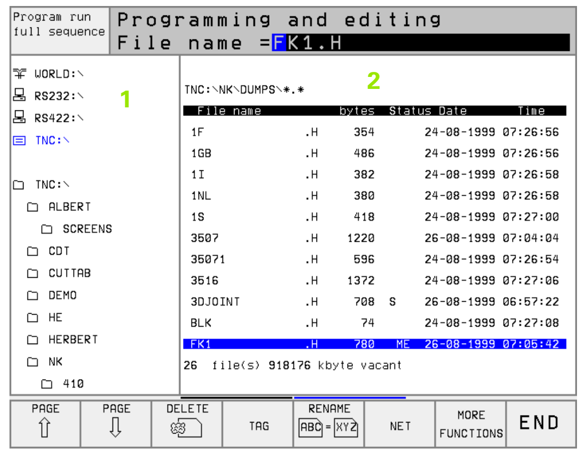
If you copy files into a directory in which other files are stored under the same name, the TNC will ask whether the files in the target directory should be overwritten:

- To overwrite all files, press the YES soft key, or
- To overwrite no files, press the NO soft key, or
- To confirm each file separately before overwriting it, press the CONFIRM soft key.

If you wish to overwrite a protected file, this must also be confirmed or aborted separately.

The TNC in a network (applies only for Ethernet interface option)

> To connect the Ethernet card to your network, (see "Ethernet Interface (not TNC 410)" on page 400).
>
> The TNC logs error messages during network operation (see "Ethernet Interface (not TNC 410)" on page 400).



If the TNC is connected to a network, the directory window **1** displays up to 7 drives (see figure at right). All the functions described above (selecting a drive, copying files, etc.) also apply to network drives, provided that you have been given the corresponding rights.

Connecting and disconnecting network drive

PGM MGT

- To select the program management: Press the PGM MGT key. If necessary, press the WINDOW soft key to set up the screen as it is shown at right.

NET

- To manage the network drives: Press the NETWORK soft key (second soft-key row). In the right-hand window **2** the TNC shows the network drives available for access. With the following soft keys you can define the connection for each drive.

Function	Soft key
Establish network connection. If the connection is active, the TNC shows an **M** in the **Mnt** column. You can connect up to 7 additional drives with the TNC.	MOUNT DEVICE
Delete network connection.	UNMOUNT DEVICE
Automatically establish network connection whenever the TNC is switched on. The TNC shows an **A** in the **Auto** column if the connection is established automatically.	AUTO MOUNT
Do not establish network connection automatically when the TNC is switched on.	NO AUTO MOUNT

It may take some time to mount a network device. At the upper right of the screen the TNC displays **[READ DIR]** to indicate that a connection is being established. The maximum data transmission rate lies between 200 and 1000 kilobaud, depending on the file type being transmitted.

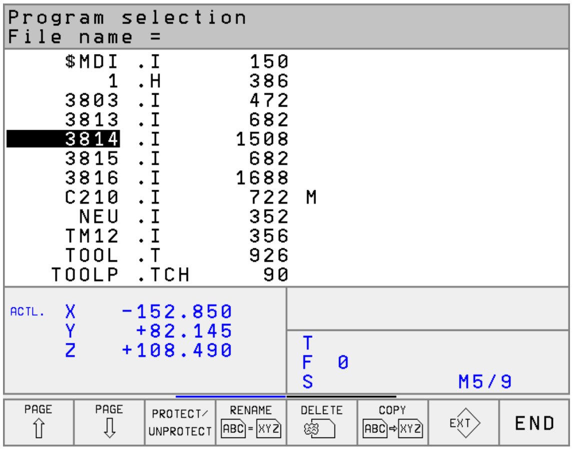
Printing file with a network printer

If you have defined a network printer (see "Ethernet Interface (not TNC 410)" on page 400), you can print the files directly:

- To call the file manager, press the PGM MGT key.
- Move the highlight to the file you wish to print.
- Press the KOPIEREN soft key.
- Press the PRINT soft key: If you have define only one printer, the TNC will print the file immediately. If you have defined more than one printer, the TNC opens a window listing all defined printers. Use the arrow keys to select the desired printer, then press ENT

4.5 File Management for the TNC 410

Calling the file manager



PGM MGT — Press the PGM MGT key: The TNC displays the file management window (see figure at right)

The window shows you all of the files that are stored in the TNC. Each file is shown with additional information:

Display	Meaning
`FILE NAME`	Name with up to 16 characters and file type
`STATUS`	File properties:
M	Program is selected in a program run operating mode.
P	File is protected against editing and erasure.

Selecting a file

PGM MGT — Call the file manager.

Use the arrow keys or the arrow soft keys to move the highlight to the file you wish to select:

↓ ↑ — Moves the highlight up or down **file by file** in the window.

PAGE ⇩ PAGE ⇧ — Moves the highlight up or down **page by page** in the window.

ENT — To select the file: Press the ENT key.

Deleting a file

PGM MGT — Call the file manager.

Use the arrow keys or the arrow soft keys to move the highlight to the file you wish to delete:

Moves the highlight up or down **file by file** in the window.

PAGE ⇩ PAGE ⇧ — Moves the highlight up or down **page by page** in the window.

DELETE — To delete the file: Press the DELETE soft key.

Delete ..... file?

YES — Confirm with the YES soft key.

NO — Abort with the NO soft key.

Copying a file

PGM MGT — Call the file manager.

Use the arrow keys or the arrow soft keys to move the highlight to the file you wish to copy:

↑ ↓ — Moves the highlight up or down **file by file** in the window.

PAGE ⇧ PAGE ⇩ — Moves the highlight up or down **page by page** in the window.

COPY ABC ⇨ XYZ — To copy the file: Press the COPY soft key.

`Target file =`

Enter the new file name and confirm your entry with the ENT key.

Data transfer to or from an external data medium

> Before you can transfer data to an external data medium, you must setup the data interface (see "Setting the Data Interface for the TNC 410" on page 393).

Call the file manager.

Activate data transfer: Press the EXT soft key. In the left half of the screen the TNC shows all files saved on its hard disk. In the right half of the screen it shows all files saved on the external data medium.

Use the arrow keys to highlight the file(s) that you want to transfer:

Moves the highlight up and down within a window.

Moves the highlight from the left to the right window, and vice versa.

If you wish to copy from the TNC to the external data medium, move the highlight in the left window to the file to be transferred.

If you wish to copy from the external data medium to the TNC, move the highlight in the right window to the file to be transferred.

> If a file to be read in already exists in the memory of the TNC, the TNC displays the message **`File xxx already exists. Read in file?`** In this case, answer the dialog question with YES (file is the read in) or NO (file is not read in).
>
> Likewise, if a file to be read out already exists on the external device, the TNC asks whether you wish to overwrite the external file.

Read in all files (file types: .H, .I, .T, .TCH, .D, .PNT)

- Read in all of the files that are stored on the external data medium.

Read in offered file

- List all files of a certain file type.
- For example: list all HEIDENHAIN conversational programs. To read-in the listed program, press the YES soft key. If you do not wish the read-in the program, press NO.

Read in a specific file

- Enter the file name. Confirm with the ENT key.
- Select the file type, e.g. HEIDENHAIN dialog program.

If you with to read-in the tool table TOOL.T, press the TOOL TABLE soft key. If you with to read-in the tool-pocket table TOOLP.TCH, press the POCKET TABLE soft key.

Read out a specific file

- Select the function for reading out a single file.
- Move the highlight to the file that you wish to read out. Press ENT or the TRANSFER soft key to start the transfer.
- To terminate the function for reading out specific files: press the END key.

Read out all files (file types: .H, .I, .T, . TCH, .D, .PNT)

- Output all files stored in the TNC to an external device.

Display a file directory of the external device (file types: .H, .I, .T, .TCH, .D, .PNT)

- Display a list of files stored in the external device. The files are displayed pagewise. To show the next page: press the YES soft key. To return to the main menu: press the NO soft key.

4.6 Creating and Writing Programs

Organization of an NC program in ISO format

A part program consists of a series of program blocks. The figure at right illustrates the elements of a block.

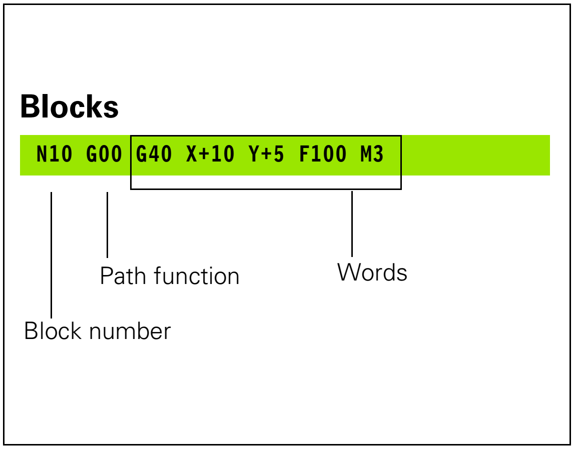
The TNC numbers the blocks in ascending sequence.

The first block of a program is identified by **%**, the program name and the active unit of measure (G70/G71).

The subsequent blocks contain information on:

- The workpiece blank
- Tool definitions, tool calls
- Feed rates and spindle speeds, as well as
- Path contours, cycles and other functions

The last block of a program is identified by **N999999, %**, the program name and the active unit of measure (G70/G71).



Define blank form: G30/G31

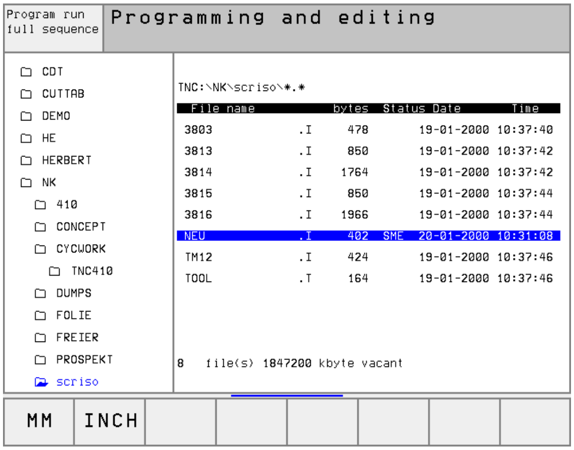
Immediately after initiating a new program, you define a cuboid workpiece blank. This definition is needed for the TNC's graphic simulation feature. The sides of the workpiece blank lie parallel to the X, Y and Z axes and may be up to 100 000 mm long (TNC 410: 30 000 mm). The blank form is defined by two of its corner points:

- MIN point G30: the smallest X, Y and Z coordinates of the blank form, entered as absolute values.
- MAX point G31: the largest X, Y and Z coordinates of the blank form, entered as absolute or incremental values (with G91).

> You only need to define the blank form if you wish to run a graphic test for the program!
>
> The TNC can display the graphic only if the ratio of the shortest to the longest side of the blank form is less than 1 : 64.

Creating a new part program TNC 426, TNC 430

You always enter a part program in the **Programming and Editing** mode of operation:



Select the **Programming and Editing** mode of operation.

PGM MGT

To call the file manager, press the PGM MGT key.

Select the directory in which you wish to store the new program:

File name = OLD.H

ENT

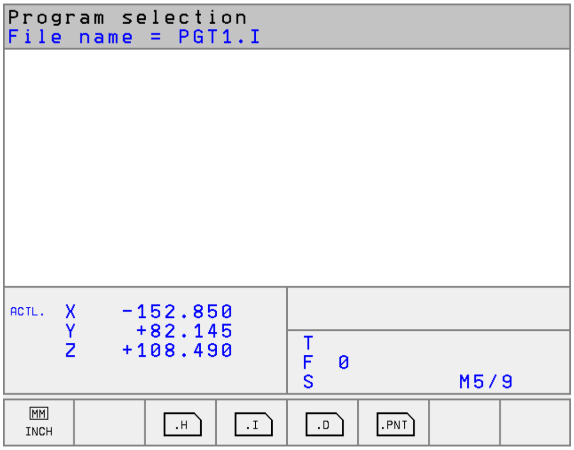
Enter the new program name and confirm your entry with the ENT key.

MM

To select the unit of measure, press the MM or INCH soft key. The TNC switches the screen layout and initiates the dialog for defining the blank form.

Creating a new part program TNC 410

You always enter a part program in the **Programming and Editing** mode of operation:



Select the **Programming and Editing** mode of operation.

PGM MGT

To call the file manager, press the PGM MGT key.

Select the directory in which you wish to store the new program:

File name = OLD.H

ENT

Enter the new program name and confirm your entry with the ENT key.

.I

Select the file type, e.g. ISO program: Press the .I soft key.

MM INCH

If necessary, switch to inches as unit of measure: Press the MM/INCH soft key.

ENT

Confirm your entry with the ENT key.

Define the workpiece blank

G 30 ENT	Define the MIN point and confirm your entry with the ENT key.

Spindle axis?

G 17 ENT	Define the spindle axis (here Z).

Def BLK FORM: Min-corner ?

X 0 ENT Y 0 ENT Z -40 ENT	Enter in sequence the X, Y and Z coordinates of the MIN point.
END	To terminate the block, press the END key.
G 31 ENT	Define the MAX point and confirm your entry with the ENT key.

Def BLK FORM: MAX-corner ?

G90 G91	Define absolute/incremental input; can be defined separately for each coordinate.
X 100 ENT Y 100 ENT Z 0 ENT	Enter in sequence the X, Y and Z coordinates of the MAX point.
END	To terminate the block, press the END key.

Example: Display the blank form in the NC program

`%NEW G71 *`	Program begin, name, unit of measure
`N10 G30 G17 X+0 Y+0 Z-40 *`	Spindle axis, MIN point coordinates
`N20 G31 G90 X+100 Y+100 Z+0 *`	MAX point coordinates
`N999999 %NEW G71 *`	Program end, name, unit of measure

The TNC automatically generates the first and last blocks of the program.

The TNC can display the graphic only if the ratio of the shortest to the longest side of the blank form is less than 1 : 64.

Programming tool movements

To program a block, select an ISO function key on the alphabetic keyboard. On the TNC 410 you can also use the gray contouring keys to get the corresponding G code.

Example of a positioning block

G 1	Start block.
G 40	Select tool movement without radius compensation.
X 10	Enter the target coordinate for the X axis.
Y 5	Enter the target coordinate for the Y axis, and go to the next question with ENT.
F 100	Enter a feed rate of 100 mm/min for this path contour.
M 3 END	Enter the miscellaneous function M3 "Spindle ON"; press the END key to terminate the block.

The program blocks window will display the following line:

```
N30 G01 G40 X+10 Y+5 F100 M3 *
```

Editing a program with TNC 426, TNC 430

While you are creating or editing a part program, you can select any desired line in the program or individual words in a block with the arrow keys or the soft keys:

Function	Soft key/key
Go to previous page	PAGE ⇧
Go to next page	PAGE ⇩
Go to beginning of program	BEGIN ⇧
Go to end of program	END ⇩
Move from one block to the next	↓ ↑
Select individual words in a block	← →

Function	Key
Set the selected word to zero	CE
Erase an incorrect number	CE
Clear a (non-blinking) error message	CE
Delete the selected word	NO ENT
Delete the selected block	DEL □
Erase cycles and program sections: First select the last block of the cycle or program section to be erased, then erase with the DEL key.	DEL □

Inserting blocks at any desired location

- Select the block after which you want to insert a new block and initiate the dialog.

Editing and inserting words

- Select a word in a block and overwrite it with the new one. The plain-language dialog is available while the word is highlighted.
- To accept the change, press the END key.

If you want to insert a word, press the horizontal arrow key repeatedly until the desired dialog appears. You can then enter the desired value.

Looking for the same words in different blocks

For this function, set the AUTO DRAW soft key to OFF.

→	To select a word in a block, press the arrow keys repeatedly until the highlight is on the desired word.
↓	Select a block with the arrow keys.

The word that is highlighted in the new block is the same as the one you selected previously.

Marking, copying, deleting and inserting program sections

The TNC provides certain functions (listed in table below) for copying program sections within an NC program or into another NC program.

To copy a program section, proceed as follows:

- Select the soft-key row using the marking function.
- Select the first (last) block of the section you wish to copy.
- To mark the first (last) block: Press the SELECT BLOCK soft key. The TNC then highlights the first character of the block and superimposes the soft key CANCEL SELECTION.
- Move the highlight to the last (first) block of the program section you wish to copy or delete. The TNC shows the marked blocks in a different color. You can end the marking function at any time by pressing the CANCEL SELECTION soft key.
- To copy the selected program section: Press the COPY BLOCK soft key, and to delete the selected section: Press the DELETE BLOCK soft key. The TNC stores the selected block.
- Using the arrow keys, select the block after which you wish to insert the copied (deleted) program section.

> To insert the section into another program, select the corresponding program using the File Manager and then mark the block after which you wish to insert the copied block.

- To insert the block: Press the INSERT BLOCK soft key.

Function	Soft key
Switch on marking function	SELECT BLOCK
Switch off marking function	CANCEL SELECTION
Delete marked block	DELETE BLOCK
Insert block that is stored in the buffer memory	INSERT BLOCK
Copy marked block	COPY BLOCK

Regenerating the block number increment

If you have deleted, moved or added program sections, you can have the TNC renumber the blocks through the ORDER N function.

- To regenerate the block numbering: Press the ORDER N soft key.
- The TNC displays the dialog prompt “Block nr. increment = .”
- Enter the desired block number increment. The value defined in MP7220 is overwritten.
- To number the blocks: Press the ENT key.
- To cancel the change: Press the END key or the END soft key.

Editing a program with TNC 410

While you are creating or editing a part program, you can select any desired line in the program or individual words in a block with the arrow keys or the soft keys. When you are entering a new block the TNC identifies the block with a * as long as the block has not been saved.

Function	Soft key/key
Go to previous page	PAGE ⇧
Go to next page	PAGE ⇩
Go to beginning of program	BEGIN ⇧
Go to end of program	END ⇩
Move from one block to the next	↓ ↑
Select individual words in a block	← →

Function	Key
Set the selected word to zero	CE
Erase an incorrect number	CE
Clear a (non-blinking) error message	CE
Delete the selected word	NO ENT
In a block: Restore previously saved version	DEL □
Delete the selected block	DEL □
Erase cycles and program sections: First select the last block of the cycle or program section to be erased, then erase with the DEL key.	DEL □

Inserting blocks at any desired location

- Select the block after which you want to insert a new block and initiate the dialog.

Editing and inserting words

- Select a word in a block and overwrite it with the new one. The plain-language dialog is available while the word is highlighted.
- To accept the change, press the END key.
- To reject the change, press the DEL key.

If you want to insert a word, press the horizontal arrow key repeatedly until the desired dialog appears. You can then enter the desired value.

Looking for the same words in different blocks

For this function, set the AUTO DRAW soft key to OFF.

→	To select a word in a block, press the arrow keys repeatedly until the highlight is on the desired word.
↓ ↑	Select a block with the arrow keys.

The word that is highlighted in the new block is the same as the one you selected previously.

Finding any text

- To select the search function, press the FIND soft key. The TNC displays the dialog prompt **`Find text:`**
- Enter the text that you wish to find.
- To find the text, press the EXECUTE soft key.

Inserting the previously edited (deleted) block at any location

- Select the block after which you want to insert the block you have just edited (deleted) and press the INSERT NC BLOCK soft key.

Block display

- If a block is so long that the TNC cannot display it in one line (for example in a fixed cycle), this will be indicated with ">>" at the right edge of the screen.

4.7 Interactive Programming Graphics (only TNC 410)

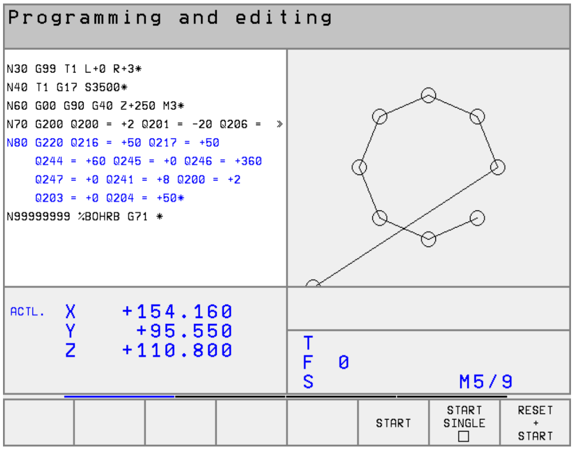
To generate/not generate graphics during programming:

While you are writing the part program, you can have the TNC generate a 2-D pencil-trace graphic of the programmed contour.

- To switch the screen layout to displaying program blocks to the left and graphics to the right, press the SPLIT SCREEN key and PGM + GRAPHICS soft key.

AUTO DRAW OFF / ON

- Set the AUTO DRAW soft key to ON. While you are entering the program lines, the TNC generates each path contour you program in the graphics window in the right screen half.



If you do not wish to have graphics generated during programming, set the AUTO DRAW soft key to OFF.

Even when AUTO DRAW ON is active, graphics are not generated for program section repeats.

Generating a graphic for an existing program

- Use the arrow keys to select the block up to which you want the graphic to be generated, or press GOTO and enter the desired block number.

RESET + START

- To generate graphics, press the RESET + START soft key.

Additional functions:

Function	Soft key
Generate a complete graphic	RESET + START
Generate interactive graphic blockwise	START SINGLE
Generate a complete graphic or complete it after RESET + START	START
Stop the programming graphics. This soft key only appears while the TNC generates the interactive graphics	STOP

Magnifying or reducing a detail

You can select the graphics display by selecting a detail with the frame overlay. You can now magnify or reduce the selected detail.

▶ Select the soft-key row for detail magnification/reduction (second row, see figure at center right).

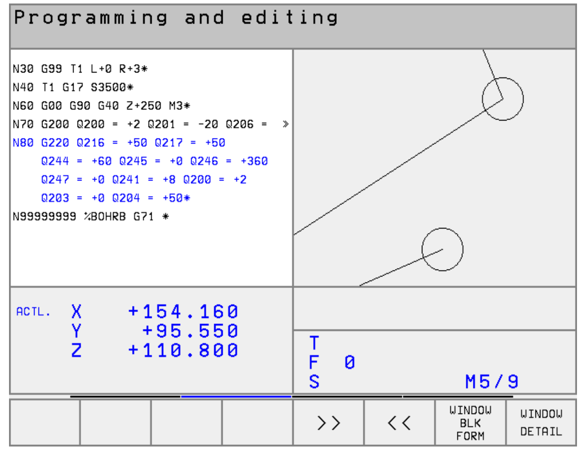
The following functions are available:

Function	Soft keys/keys
Reduce the frame overlay — press and hold the soft key to reduce the detail.	<<
Enlarge the frame overlay—press and hold the soft key to magnify the detail.	>>
Shift the frame overlay. Press and hold the desired key to move the frame overlay.	← → ↓ ↑

WINDOW DETAIL

▶ Confirm the selected area with the WINDOW DETAIL soft key.

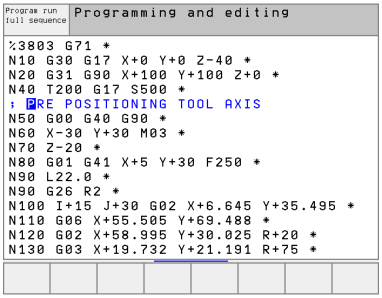
With the WINDOW BLK FORM soft key, you can restore the original section.

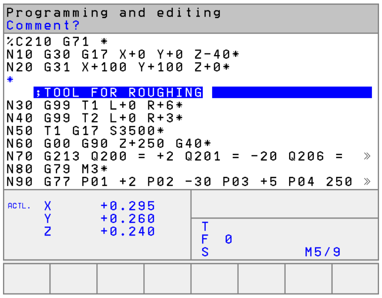


4.8 Adding Comments

Function

You can add comments to any desired block in the part program to explain program steps or make general notes. There are three possibilities to add comments:





Adding comments during program input (not TNC 410)

- ▶ Enter the data for a program block, then press the semicolon key ";" on the alphabetic keyboard—the TNC displays the dialog prompt **COMMENT ?**
- ▶ Enter your comment and conclude the block by pressing the END key.

Adding comments after program input (not TNC 410)

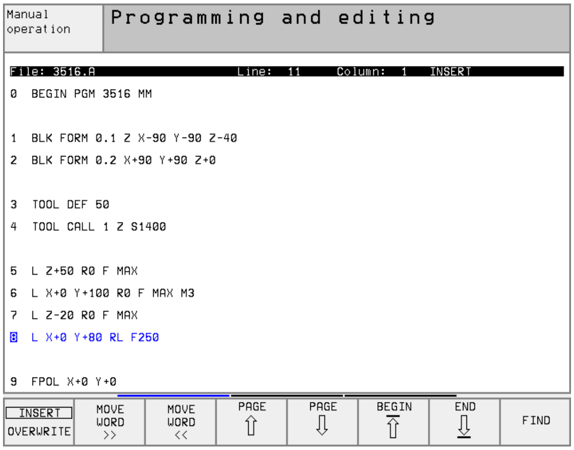
- ▶ Select the block to which a comment is to be added.
- ▶ Select the last word in the block with the right arrow key: A semicolon appears at the end of the block and the TNC displays the dialog prompt **COMMENT ?**
- ▶ Enter your comment and conclude the block by pressing the END key.

Entering a comment in a separate block

- ▶ Select the block after which the comment is to be inserted.
- ▶ Initiate the programming dialog with the semicolon key ";" on the alphabetic keyboard.
- ▶ Enter your comment and conclude the block by pressing the END key.

4.9 Creating Text Files (not TNC 410)

Function



You can use the TNC's text editor to write and edit texts. Typical applications:

- Recording test results
- Documenting working procedures
- Creating formularies

Text files are type .A files (ASCII files). If you want to edit other types of files, you must first convert them into type .A files.

Opening and exiting text files

- ▶ Select the Programming and Editing mode of operation.
- ▶ To call the file manager, press the PGM MGT key.
- ▶ To display type .A files, press the SELECT TYPE and then the SHOW .A soft keys.
- ▶ Select a file and open it with the SELECT soft key or ENT key, or create a new file by entering the new file name and confirming your entry with the ENT key.

To leave the text editor, call the file manager and select a file of a different file type, for example a part program.

Cursor movements	Soft key
Move one word to the right	MOVE WORD >>
Move one word to the left	MOVE WORD <<
Go to next screen page	PAGE ⇩
Go to previous screen page	PAGE ⇧
Go to beginning of file	BEGIN ⇧
Go to end of file	END ⇩

Editing functions	Key
Begin a new line	RET

Editing functions	Key
Erase the character to the left of the cursor	X
Insert a blank space	SPACE
Switch between upper and lower case letters	SHIFT SPACE

Editing texts

The first line of the text editor is an information headline which displays the file name, and the location and writing mode of the cursor:

File: Name of the text file
Line: Line in which the cursor is presently located
Column: Column in which the cursor is presently located
INSERT: Insert new text, pushing the existing text to the right
OVERWRITE: Write over the existing text, erasing it where it is replaced with the new text.

The text is inserted or overwritten at the location of the cursor. You can move the cursor to any desired position in the text file by pressing the arrow keys.

The line in which the cursor is presently located is depicted in a different color. A line can have up to 77 characters. To start a new line, press the RET key or the ENT key.

Erasing and inserting characters, words and lines

With the text editor, you can erase words and even lines, and insert them at any desired location in the text.

- Move the cursor to the word or line you wish to erase and insert at a different place in the text.
- Press the DELETE WORD or DELETE LINE soft key: The text is placed in the buffer memory.
- Move the cursor to the location where you wish insert the text, and press the RESTORE LINE/WORD soft key.

Function	Soft key
Delete and temporarily store a line	DELETE LINE
Delete and temporarily store a word	DELETE WORD
Delete and temporarily store a character	DELETE CHAR
Insert a line or word from temporary storage	INSERT LINE / WORD

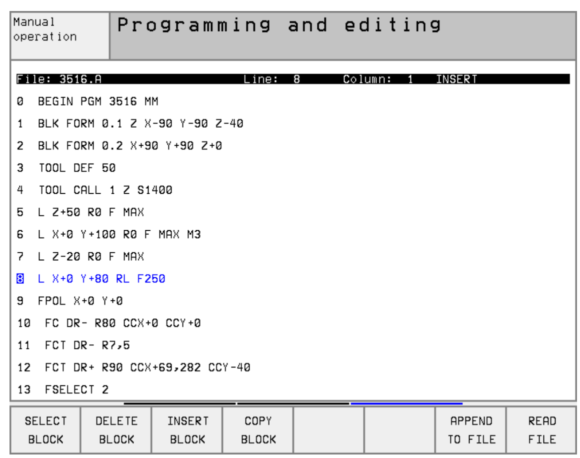
Editing text blocks

You can copy and erase text blocks of any size, and insert them at other locations. Before carrying out any of these editing functions, you must first select the desired text block:

- To select a text block, move the cursor to the first character of the text you wish to select.

SELECT BLOCK

- Press the SELECT BLOCK soft key.
- Move the cursor to the last character of the text you wish to select. You can select whole lines by moving the cursor up or down directly with the arrow keys—the selected text is shown in a different color.



After selecting the desired text block, you can edit the text with the following soft keys:

Function	Soft key
Delete the selected text and store temporarily	DELETE BLOCK
Store marked block temporarily without erasing (copy)	INSERT BLOCK

If necessary, you can now insert the temporarily stored block at a different location:

- Move the cursor to the location where you want to insert the temporarily stored text block.

INSERT BLOCK

- Press the INSERT BLOCK soft key for the text block to be inserted.

You can insert the temporarily stored text block as often as desired.

To transfer the selected text to a different file

- Select the text block as described previously.

APPEND TO FILE

- Press the APPEND TO FILE soft key. The TNC displays the dialog prompt **Destination file =**
- Enter the path and name of the target file. The TNC appends the selected text to the end of the specified file. If no target file with the specified name is found, the TNC creates a new file with the selected text.

To insert another file at the cursor position

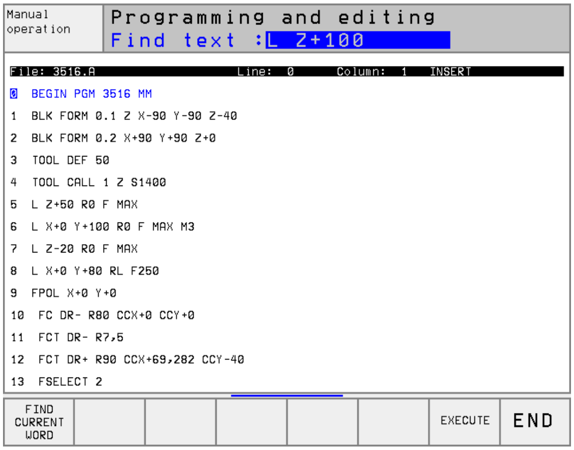
- Move the cursor to the location in the text where you wish to insert another file.

READ FILE

- Press the READ FILE soft key. The TNC displays the dialog prompt **File name =**
- Enter the path and name of the file you want to insert.

Finding text sections

With the text editor, you can search for words or character strings in a text. Two functions are available:



Finding the current text

The search function is to find the next occurrence of the word in which the cursor is presently located:

- Move the cursor to the desired word.
- To select the search function, press the FIND soft key.
- Press the FIND CURRENT WORD soft key.
- To leave the search function, press the END soft key.

Finding any text

- To select the search function, press the FIND soft key. The TNC displays the dialog prompt `Find text:`
- Enter the text that you wish to find.
- To find the text, press the EXECUTE soft key.
- To leave the search function, press the END soft key.

4.10 Integrated Pocket Calculator (not TNC 410)

Operation

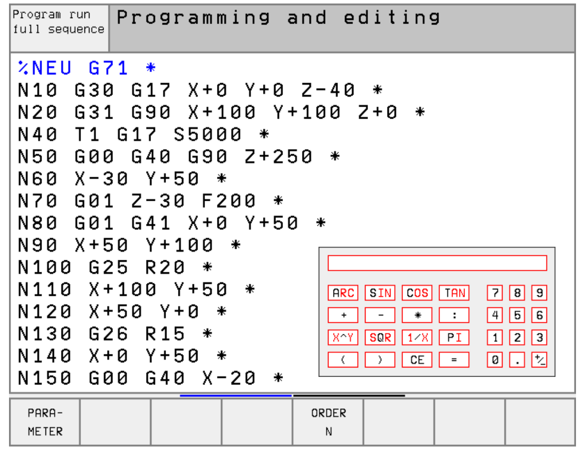
The TNC features an integrated pocket calculator with the basic mathematical functions.

With the CALC key you can open and close an additional window for calculations. You can move the window to any desired location on the TNC screen with the arrow keys.

The calculator is operated with short commands through the alphabetic keyboard. The commands are shown in a special color in the calculator window:

Mathematical function	Command (key)
Addition	+
Subtraction	–
Multiplication	*
Division	:
Sine	S
Cosine	C
Tangent	T
Arc sine	AS
Arc cosine	AC
Arc tangent	AT
Powers	^
Square root	Q
Inversion	/
Parenthetic calculations	( )
p (3.14159265359)	P
Display result	=

If you are writing a program and the programming dialog is active, you can use the actual-position-capture key to transfer the result to the highlight position in the current block.



4.11 Direct Help for NC Error Messages (not TNC 410)

Displaying error messages

The TNC automatically generates error messages when it detects problems such as

- Incorrect data input
- Logical errors in the program
- Contour elements that are impossible to machine
- Incorrect use of the touch probe system

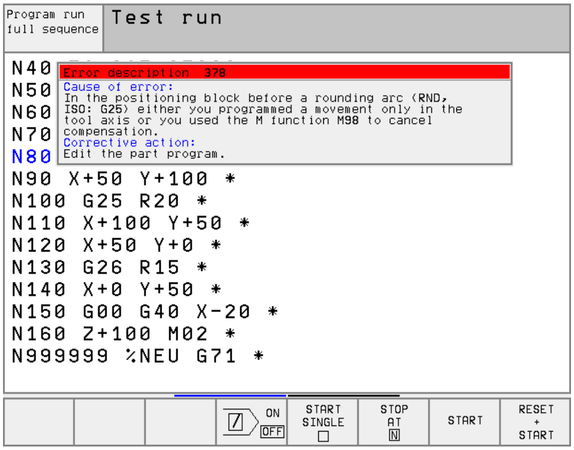
An error message that contains a program block number was caused by an error in the indicated block or in the preceding block. The TNC error messages can be canceled with the CE key, after the cause of the error has been removed.

If you require more information on a particular error message, press the HELP key. A window is then superimposed where the cause of the error is explained and suggestions are made for correcting the error.

Display HELP

- ▶ To display Help, press the HELP key.
- ▶ Read the description of the error and the possibilities for correcting it. Close the Help window with the CE key, thus canceling the error message.
- ▶ Remove the cause of the error as described in the Help window.

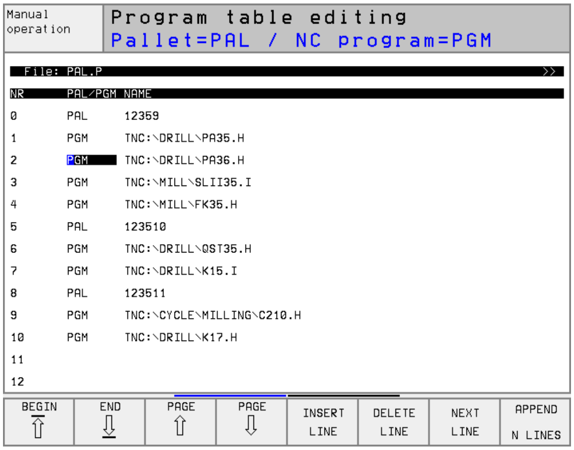
The TNC displays the Help text automatically if the error message is flashing. The TNC needs to be restarted after blinking error messages. To restart the TNC, press the END key and hold for two seconds.



4.12 Pallet Management (not TNC 410)

Function

Pallet table management is a machine-dependent function. The standard functional range will be described below. Refer to your machine manual for more information.



Pallet tables are used for machining centers with pallet changer: The pallet table calls the part programs that are required for the different pallets, and activates datum shifts or datum tables.

You can also use pallet tables to run in succession several programs that have different datums.

Pallet tables contain the following information:

- **PAL/PGM** (entry obligatory):
  Identification for pallet or NC program (select with ENT or NO ENT)
- **NAME** (entry obligatory):
  Pallet or program name. The machine tool builder determines the pallet name (see Machine Manual). The program name must be stored in the same directory as the pallet table. Otherwise you must enter the full path name for the program.
- **DATUM** (entry optional):
  Name of the datum table. The datum table must be stored in the same directory as the pallet table. Otherwise you must enter the full path name for the datum table. Datums from the datum table can be activated in the NC program with Cycle **G53 DATUM SHIFT**.
- **X, Y, Z** (entry optional, other axes also possible):
  For pallet names, the programmed coordinates are referenced to the machine datum. For NC programs, the programmed coordinates are referenced to the pallet datum. These entries overwrite the datum that you last set in the Manual mode of operation. With the miscellaneous function M104 you can reactivate the datum that was last set. With the actual-position-capture key, the TNC opens a window that enables you to have the TNC enter various points as datums (see table below):

Position	Meaning
Actual values	Enter the coordinates of the current tool position relative to the active coordinate system.
Reference values	Enter the coordinates of the current tool position relative to the machine datum.
ACTUAL measured values	Enter the coordinates relative to the active coordinate system of the datum last probed in the Manual operating mode.
REF measured values	Enter the coordinates relative to the machine datum of the datum last probed in the Manual operating mode.

With the arrow keys and ENT, select the position that you wish to confirm. Then press the ALL VALUES soft key so that the TNC saves the respective coordinates of all active axes in the pallet table. With the PRESENT VALUE soft key, the TNC saves the coordinates of the axis on which the highlight in the pallet table is presently located.

> ☞ If you have not defined a pallet before an NC program, the programmed coordinates are then referenced to the machine datum. If you do not define an entry, the datum that was set manually remains active.

Editing function	Soft key
Select beginning of table	BEGIN ⇧
Select end of table	END ⇩
Select previous page in table	PAGE ⇧
Select next page in table	PAGE ⇩
Insert the last line in the table	INSERT LINE
Delete the last line in the table	DELETE LINE
Go to the beginning of the next line	NEXT LINE
Add the entered number of lines to the end of the table	APPEND N LINES
Copy the highlighted field (2nd soft-key row)	COPY FIELD
Insert the copied field (2nd soft-key row)	PASTE FIELD

Selecting a pallet table

- Call the file manager in the Programming and Editing or Program Run mode: Press the PGM MGT key.
- Display all type .P files: Press the soft keys SELECT TYPE and SHOW .P.
- Select a pallet table with the arrow keys, or enter a new file name to create a new table.
- Confirm your entry with the ENT key.

Leaving the pallet file

- To call the file manager, press the PGM MGT soft key.
- To select a different type of file, press the SELECT TYPE soft key and the soft key for the desired file type, for example SHOW.H.
- Select the desired file.

Executing the pallet file

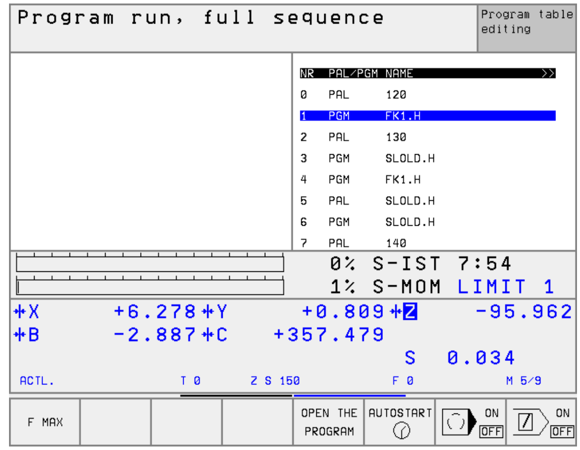
> In machine parameter 7683, set whether the pallet table is to be executed blockwise or continuously (see “General User Parameters” on page 422).

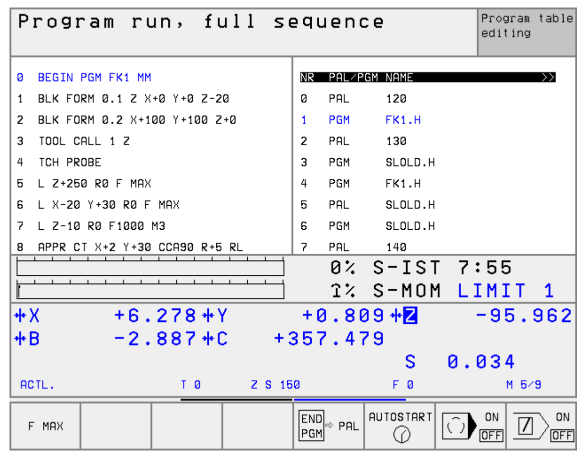
- Select the file manager in the operating mode Program Run, Full Sequence or Program Run, Single Block: Press the PGM MGT key.
- Display all type .P files: Press the soft keys SELECT TYPE and SHOW .P.
- Select pallet table with the arrow keys and confirm with ENT.
- To execute pallet table: Press the NC Start button. The TNC executes the pallets as set in Machine Parameter 7683.

Screen layout for executing pallet tables

You can have the TNC display the program contents and pallet file contents on the screen together by selecting the screen layout PGM + PALLET. During execution, the TNC then shows program blocks to the left and the pallet to the right. To check the program contents before execution, proceed as follows:

- Select a pallet table.
- With the arrow keys, choose the program you would like to check.
- Press the OPEN PGM soft key: The TNC displays the selected program on the screen. You can now page through the program with the arrow keys.
- To return to the pallet table, press the END PGM soft key.



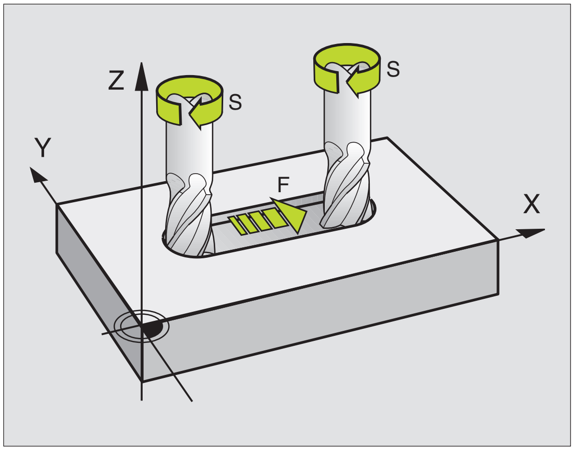


5

Programming: Tools

5.1 Entering Tool-Related Data

Feed rate F



The feed rate **F** is the speed (in millimeters per minute or inches per minute) at which the tool center moves. The maximum feed rates can be different for the individual axes and are set in machine parameters.

Input

You can enter the feed rate in every positioning block or in a separate block. Press the F key on the alphabetic keyboard.

Rapid traverse

If you wish to program rapid traverse, enter **F MAX.** To enter **F MAX,** press the ENT key or the F MAX soft key when the dialog question **FEED RATE F = ?** appears on the TNC screen.

Duration of effect

A feed rate entered as a numerical value remains in effect until a block with a different feed rate is reached. If the new feed rate is **G00** (rapid traverse), the last programmed feed rate is once again valid after the next block with **G01**.

Changing during program run

You can adjust the feed rate during program run with the feed-rate override knob.

Spindle speed S

The spindle speed S is entered in revolutions per minute (rpm) in any block (e.g. during tool call).

Programmed change

In the part program, you can change the spindle speed with an S block:

S

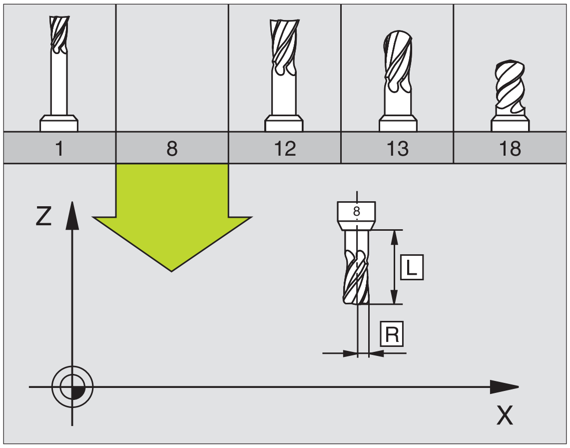
- Press the S key on the alphabetic keyboard.
- Enter the new spindle speed.

Changing during program run

You can adjust the spindle speed during program run with the spindle-speed override knob.

5.2 Tool Data

Requirements for tool compensation



You usually program the coordinates of path contours as they are dimensioned in the workpiece drawing. To allow the TNC to calculate the tool center path—i.e. the tool compensation—you must also enter the length and radius of each tool you are using.

Tool data can be entered either directly in the part program with **G99** or separately in tool tables. In a tool table, you can also enter additional data on the specific tool. The TNC will consider all the data entered for the tool when executing the part program.

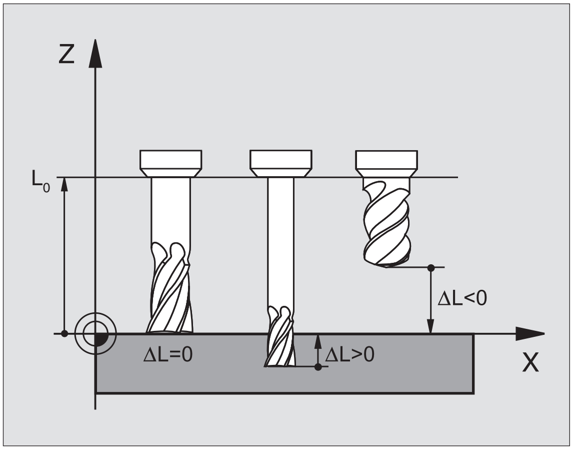
Tool numbers and tool names

Each tool is identified by a number between 0 and 254. If you are working with tool tables, you can use higher numbers and you can also enter a tool name for each tool (not TNC 410).

The tool number 0 is automatically defined as the zero tool with the length L=0 and the radius R=0.

In tool tables, tool 0 should also be defined with L=0 and R=0.

Tool length L



There are two ways to determine the tool length L:

Determining the difference between the length of the tool and that of a zero tool L0

For the algebraic sign:

L>L0:	The tool is longer than the zero tool
L<L0:	The tool is shorter than the zero tool

To determine the length:

- Move the zero tool to the reference position in the tool axis (e.g. workpiece surface with Z=0).
- Set the datum in the tool axis to 0 (datum setting).
- Insert the desired tool.
- Move the tool to the same reference position as the zero tool.
- The TNC displays the difference between the current tool and the zero tool.
- Using the key for “actual position capture” (TNC 426 B, TNC 430) or the soft key ACT. POS. Z (TNC 410), transfer the value to the G99 block or the tool table.

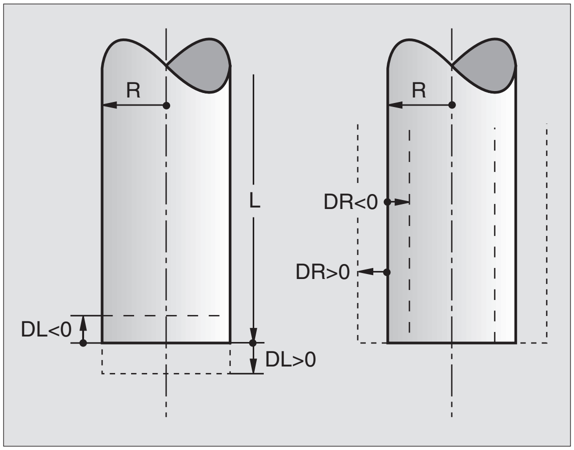
Determining the length L with a tool presetter

Enter the determined value directly in the **G99** tool definition block or in the tool table without further calculations.

Tool radius R

You can enter the tool radius R directly.

Delta values for lengths and radii



Delta values are offsets in the length and radius of a tool.

A positive delta value describes a tool oversize (DL, DR>0). If you are programming the machining data with an allowance, enter the oversize value with **T**.

A negative delta value describes a tool undersize (DL, DR<0). An undersize is entered in the tool table for wear.

Delta values are usually entered as numerical values. In a **T** block, you can also assign the values to Q parameters.

Input range: You can enter a delta value with up to ± 99.999 mm.

Entering tool data into the program

The number, length and radius of a specific tool is defined in the **G99** block of the part program.

G 99

- Select tool definition. Confirm your entry with the ENT key.
- Enter the Tool number: Each tool is uniquely identified by its number.
- Enter the tool length: Enter the compensation value for the tool length.
- Enter the Tool radius.

In the programming dialog, you can transfer the value for tool length directly into the input line.

TNC 426, TNC 430:

Press the actual-position-capture key. You only need to make sure that the highlight in the status display is placed on the tool axis.

TNC 410:

Press the ACT. POS. Z soft key.

Resulting NC block:

```
N40 G99 T5 L+10 R+5 *
```

Entering tool data in tables

You can define and store up to 32 767 tools and their tool data in a tool table (TNC 410: 254 tools). In Machine Parameter 7260, you can define how many tools are to be stored by the TNC when a new table is set up. See also the Editing Functions at a later stage in this Chapter. In order to be able to assign various compensation data to a tool (indexing tool number), machine parameter 7262 must not be equal to 0 (not TNC 410).

You must use tool tables if

- you wish to use indexed tools such as stepped drills with more than one length compensation value,
- your machine tool has an automatic tool changer,
- you want to measure tools automatically with the TT 130 touch probe (see the new Touch Probe Cycles User's Manual, Chapter 4),
- you want to rough-mill the contour with Cycle **G122** (see “ROUGH-OUT (Cycle G122)” on page 272),

Tool table: Standard tool data

Abbr.	Input	Dialog
T	Number by which the tool is called in the program (e.g. 5, indexed: 5.2)	–
NAME	Name by which the tool is called in the program	**Tool name?**
L	Value for tool length compensation L	**Tool length?**
R	Compensation value for the tool radius R	**Tool radius R?**
R2	Tool radius R2 for toroid cutters (only for 3-D radius compensation or graphical representation of a machining operation with spherical or toroid cutters)	**Tool radius R2?**
DL	Delta value for tool radius R2	**Tool length oversize?**
DR	Delta value for tool radius R	**Tool radius oversize R?**
DR2	Delta value for tool radius R2	**Tool radius oversize R2?**
LCUTS	Tooth length of the tool for Cycle 22	**Tooth length in the tool axis?**
ANGLE	Maximum plunge angle of the tool for reciprocating plunge-cut in Cycles 22 and 208	**Maximum plunge angle?**
TL	Set tool lock (**TL**: **T**ool **L**ocked)	**Tool locked?** **Yes = ENT / No = NO ENT**
RT	Number of a replacement tool (**RT**), if available (see also TIME2)	**Replacement tool?**
TIME1	Maximum tool life in minutes. This function can vary depending on the individual machine tool. Your machine manual provides more information on TIME1.	**Maximum tool age?**
TIME2	Maximum tool life in minutes during a tool call: If the current tool age exceeds this value, the TNC changes the tool during the next tool call (see also CUR.TIME).	**Maximum tool age for TOOL CALL?**

Abbr.	Input	Dialog
`CUR.TIME`	Time in minutes the tool has been in use: The TNC automatically counts the current tool age. A starting value can be entered for used tools.	**`Current tool life?`**
`DOC`	Comment on tool (up to 16 characters)	**`Tool description?`**
`PLC`	Information on this tool that is to be sent to the PLC	**`PLC status?`**
`PLC VAL`	**Only TNC 426, TNC 430:** Value of this tool that is to be sent to the PLC	**`PLC value?`**

Tool table: Tool data required for automatic tool measurement

For a description of the cycles governing automatic tool measurement, see the new Touch Probe Cycles Manual, Chapter 4.

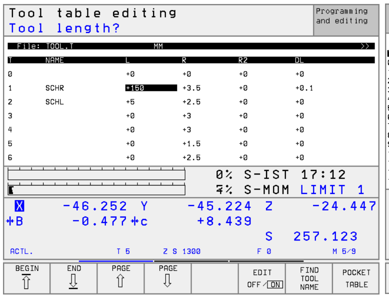
Abbr.	Input	Dialog
`CUT`	Number of teeth (20 teeth maximum)	**`Number of teeth ?`**
`LTOL`	Permissible deviation from tool length L for wear detection. If the entered value is exceeded, the TNC locks the tool (status **L**). Input range: 0 to 0.9999 mm	**`Wear tolerance: length ?`**
`RTOL`	Permissible deviation from tool radius R for wear detection. If the entered value is exceeded, the TNC locks the tool (status **L**). Input range: 0 to 0.9999 mm	**`Wear tolerance: radius ?`**
`DIRECT.`	Cutting direction of the tool for measuring the tool during rotation	**`Cutting direction (M3 = –) ?`**
`TT:R-OFFS`	For tool length measurement: tool offset between stylus center and tool center. Preset value: Tool radius R (NO ENT means **R**).	**`Tool offset: radius ?`**
`TT:L-OFFS`	Tool radius measurement: tool offset in addition to MP6530 (see "General User Parameters" on page 422) between upper surface of stylus and lower surface of tool. Default: 0	**`Tool offset: length ?`**
`LBREAK`	Permissible deviation from tool length L for breakage detection. If the entered value is exceeded, the TNC locks the tool (status **L**). Input range: 0 to 0.9999 mm	**`Breakage tolerance: length ?`**
`RBREAK`	Permissible deviation from tool radius R for breakage detection. If the entered value is exceeded, the TNC locks the tool (status L). Input range: 0 to 0.9999 mm	**`Breakage tolerance: radius ?`**

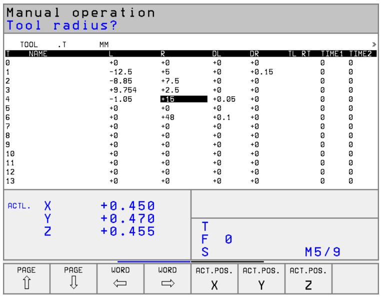
Tool table: Tool data for 3-D touch trigger probe (only when bit 1 is set in MP7411 = 1, see also the Touch Probe Cycles Manual)

Abbr.	Input	Dialog
`CAL-OF1`	During calibration, the TNC stores in this column the center misalignment in the reference axis of the 3-D probe, if a tool number is indicated in the calibration menu.	`Center misalignmt. in ref. axis?`
`CAL-OF2`	During calibration, the TNC stores in this column the center misalignment in the minor axis of the 3-D probe, if a tool number is indicated in the calibration menu.	`Center misalignment minor axis?`
`CAL-ANG`	During calibration, the TNC stores in this column the spindle angle at which the 3-D probe was calibrated, if a tool number is indicated in the calibration menu.	`Spindle angle for calibration?`

Editing tool tables

The tool table that is active during execution of the part program is designated as TOOL.T. You can only edit TOOL.T in one of the machine operating modes. Other tool tables that are used for archiving or test runs are given different file names with the extension .T.





To open the tool table TOOL.T:

- Select any machine operating mode.

TOOL TABLE

- To select the tool table, press the TOOL TABLE soft key.

EDIT OFF / ON

- Set the EDIT soft key to ON.

To open any other tool table:

- Select the Programming and Editing mode of operation.

PGM MGT

- Call the file manager.
- To select the file type, press the SELECT TYPE soft key.
- To show type .T files, press the SHOW .T soft key.
- Select a file or enter a new file name. Conclude your entry with the ENT key or the SELECT soft key.

When you have opened the tool table, you can edit the tool data by moving the cursor to the desired position in the table with the arrow keys or the soft keys. You can overwrite the stored values, or enter new values at any position. The available editing functions are illustrated in the table below.

If the TNC cannot show all positions in the tool table in one screen page, the highlight bar at the top of the table will display the symbol “**>>**” or “**<<**”.

Leaving the tool table

- Call the file manager and select a file of a different type, such as a part program.

Editing functions for tool tables TNC 426, TNC 430	Soft key
Select beginning of table	BEGIN ⇧
Select end of table	END ⇩
Select previous page in table	PAGE ⇧
Select next page in table	PAGE ⇩
Look for the tool name in the table	FIND TOOL NAME
Show tool information in columns or show all information on one tool on one screen page	LIST FORMULAR
Move to beginning of line.	BEGIN LINE ⇦
Move to end of line.	END LINE ⇨
Copy highlighted field.	COPY FIELD
Insert copied field.	PASTE FIELD
Add the entered number of lines (tools) to the end of the table.	APPEND N LINES
Insert a line for the indexed tool number after the active line. The function is only active if you are permitted to store various compensation data for a tool (machine parameter 7262 not equal to 0). The TNC inserts a copy of the tool data after the last available index and increases the index by 1. Application: e.g. stepped drill with more than one length compensation value.	INSERT LINE
Delete current line (tool).	DELETE LINE
Display / Do not display pocket numbers.	SHOW OMIT POCKET NR
Display all tools / only those tools that are stored in the pocket table.	HIDE TOOLS OFF / ON

Editing functions for tool tables TNC 410	Soft key
Select previous page in table	PAGE ⇧
Select next page in table	PAGE ⇩
Move highlight to the left	WORT ⇦
Move highlight to the right	WORT ⇨
Lock tool in TL column	YES
Do not lock tool in TL column	NO
Confirm actual position, e.g. for Z axis	AKT. POS. Z
Confirm entered value, select next column in the table	ENT
Delete incorrect value, restore previous value	CE
Restore last value stored	DEL □

Additional notes on tool tables

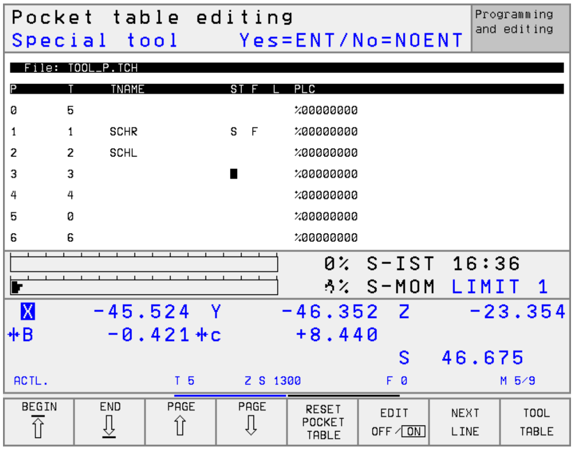
Machine parameter 7266.x defines which data can be entered in the tool table and in what sequence the data is displayed.

You can overwrite individual columns or lines of a tool table with the contents of another file. Prerequisites:

- The target file must exist.
- The file to be copied must contain only the columns (or lines) you want to replace.

To copy individual columns or lines, press the REPLACE FIELDS soft key (see "Copying a single file" on page 58).

Pocket table for tool changer



For automatic tool changing you need the pocket table TOOL_P.TCH. The TNC can manage several pocket tables with any file names. To activate a specific pocket table for program run you must select it in the file management of a Program Run mode of operation (status M).

Editing a pocket table in a Program Run operating mode

TOOL TABLE

- To select the tool table, press the TOOL TABLE soft key.

POCKET TABLE

- To select the pocket table, press the POCKET TABLE soft key.

EDIT OFF/ON

- Set the EDIT soft key to ON.

Selecting a pocket table in the Programming and operating mode (only TNC 426, TNC 430)

PGM MGT

- Call the file manager.
- To select the file type, press the SELECT TYPE soft key.
- To show files of the type .TCH, press the soft key TCH FILES (second soft-key row).
- Select a file or enter a new file name. Conclude your entry with the ENT key or the SELECT soft key.

Abbr.	Input	Dialog
P	Pocket number of the tool in the tool magazine	–
T	Tool number	**Tool number ?**
ST	Special tool with a large radius requiring several pockets in the tool magazine. If your special tool takes up pockets in front of and behind its actual pocket, these additional pockets need to be locked in column L (status L).	**Special tool ?**
F	Fixed tool number. The tool is always returned to the same pocket in the tool magazine	**Fixed pocket? Yes = ENT / No = NO ENT**
L	Locked pocket (see also column ST)	**Pocket locked Yes = ENT / No = NO ENT**
PLC	Information on this tool pocket that is to be sent to the PLC	**PLC status?**
TNAME	Display of the tool name from TOOL.T	–
DOC	Display of the comment to the tool from TOOL.T	–

Editing functions for pocket tables	Soft key
Select beginning of table	BEGIN ⇧
Select end of table	END ⇩

Editing functions for pocket tables	Soft key
Select previous page in table	PAGE ⇧
Select next page in table	PAGE ⇩
Reset pocket table	RESET POCKET TABLE
Go to the beginning of the next line	NEXT LINE
Reset tool number column T	RESET COLUMN T
Move to end of line.	END LINE ⇨

Calling tool data

A tool call in the machining program is triggered with the function T:

T 1

- **Tool number**: Enter the number of the tool. The tool must already be defined in a G99 block or in the tool table.
  For the TNC 430, TNC 430:
  To call a tool by the tool name, enter the name in quotation marks. The tool name always refers to the entry in the active tool table TOOL .T. If you wish to call a tool with other compensation values, enter also the index you defined in the tool table after the decimal point.
- **Tool length oversize DL:** Enter the delta value for the tool length.
- **Tool radius oversize DR:** Enter the delta value for the tool radius.

In a tool call, you can also program the spindle axis and feed rate, as required:

G 17

- Select spindle axis, e.g. Z axis

S 2500

- Select rotational speed and end the block with the END key

Example: Tool call

Call tool number 5 in the tool axis Z with a spindle speed 2500 rpm. The tool length is to be programmed with an oversize of 0.2 mm, the tool radius with an undersize of 1 mm.

```
N20 T 5.2 G17 S2500 DL+0.2 DR-1
```

The character **D** preceding **L** and **R** designates delta values.

Tool preselection with tool tables

When you use tool tables, enter a **G51** block to preselect the next tool to be selected. Simply enter the tool number or a corresponding Q parameter, or type the tool name in quotation marks (not TNC 410).

Tool change

The tool change function can vary depending on the individual machine tool. The machine tool manual provides further information.

Tool change position

A tool change position must be approachable without collision. With the miscellaneous functions **M91** and **M92**, you can enter machine-referenced (rather than workpiece-referenced) coordinates for the tool change position. If **T0** is programmed before the first tool call, the TNC moves the tool spindle in the tool axis to a position that is independent of the tool length.

Manual tool change

To change the tool manually, stop the spindle and move the tool to the tool change position:

- Move to the tool change position under program control.
- Interrupt program run, see "Interrupting machining," page 377.
- Change the tool.
- Resume the program run, see "Resuming program run after an interruption," page 379.

Automatic tool change

If your machine tool has automatic tool changing capability, the program run is not interrupted. When the TNC reaches a tool call with **T**, it replaces the inserted tool by another from the tool magazine.

Automatic tool change if the tool life expires: M101

The function of **M101** can vary depending on the individual machine tool. The machine tool manual provides further information.

The TNC automatically changes the tool if the tool life **TIME2** expires during program run. To use this miscellaneous function, activate **M101** at the beginning of the program. **M101** is reset with **M102**.

The tool is not always changed immediately, but, depending on the workload of the control, a few NC blocks later.

Prerequisites for standard NC blocks with radius compensation R0, RR, RL

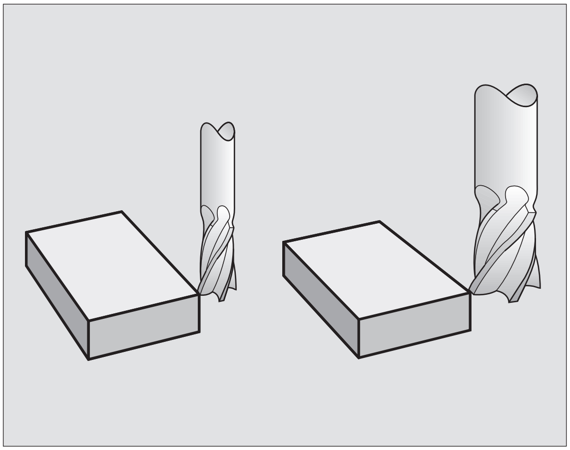
The radius of the replacement tool must be the same as that of the original tool. If the radii are not equal, the TNC displays an error message and does not replace the tool.

5.3 Tool Compensation

Introduction

The TNC adjusts the spindle path in the tool axis by the compensation value for the tool length. In the working plane, it compensates the tool radius.

If you are writing the part program directly on the TNC, the tool radius compensation is effective only in the working plane. The TNC accounts for the compensation value in up to five axes including the rotary axes.



Tool length compensation

Length compensation becomes effective automatically as soon as a tool is called and the tool axis moves. To cancel length compensation, call a tool with the length L=0.

> If you cancel a positive length compensation with **T0**, the distance between tool and workpiece will be reduced.
>
> After a tool call, the path of the tool in the tool axis, as entered in the part program, is adjusted by the difference between the length of the previous tool and that of the new one.

For tool length compensation, the TNC takes the delta values from both the **T** block and the tool table into account.

Compensation value = **L** + $\mathbf{DL}_{T}$ + $\mathbf{DL}_{TAB}$, where

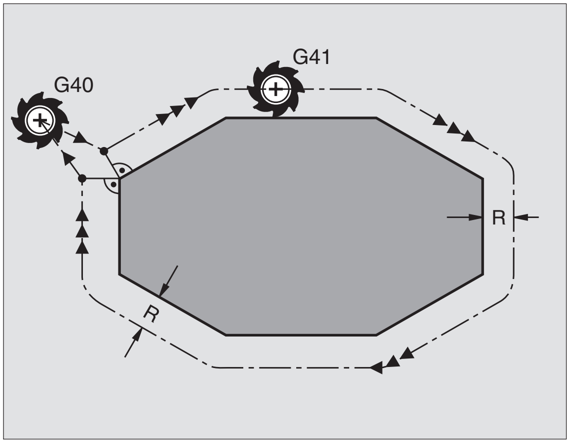
L	is the tool length **L** from the **G99** block or tool table
$\mathbf{DL}_{TL}$	is the oversize for length **DL** in the **T** block (not taken into account by the position display)
$\mathbf{DL}_{TAB}$	is the oversize for length **DL** in the tool table

Tool radius compensation

The NC block for programming a tool movement contains:

- **G41** or **G42** for radius compensation,
- **G43** or **G44**, for radius compensation with axis-parallel traverse,
- **G40**, if there is no radius compensation.

Radius compensation becomes effective as soon as a tool is called and is moved in the working plane with G41 or G42.



> The TNC automatically cancels radius compensation if you:
> - program a positioning block with **G40,**
> - program a program call with **%...,**
> - select a new program with PGM MGT.

For tool radius compensation, the TNC takes the delta values from both the **T** block and the tool table into account.

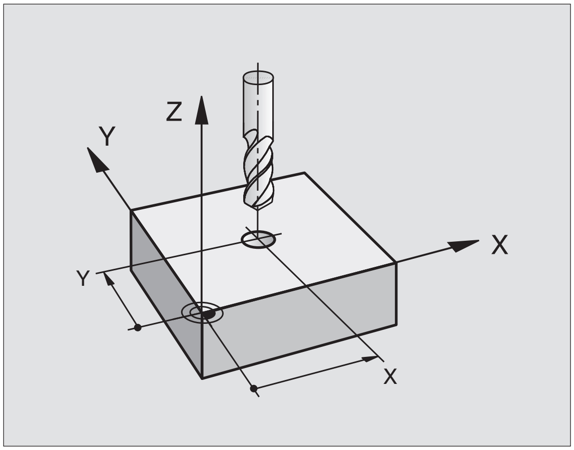
Compensation value = **R** + **DR**$_T$ + **DR**$_{TAB}$, where

R	is the tool radius **R** from the **G99** block or tool table
DR $_T$	is the oversize for radius **DR** in the **T** block (not taken into account by the position display)
DR $_{TAB}$	is the oversize for radius **DR** in the tool table

Contouring without radius compensation: R0

The tool center moves in the working plane along the programmed path or to coordinates.

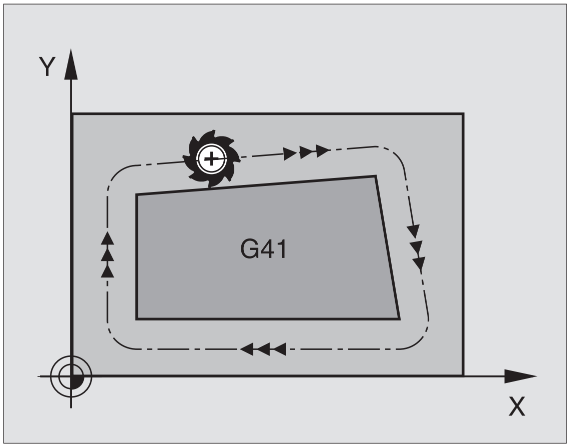
Applications: Drilling and boring, pre-positioning.

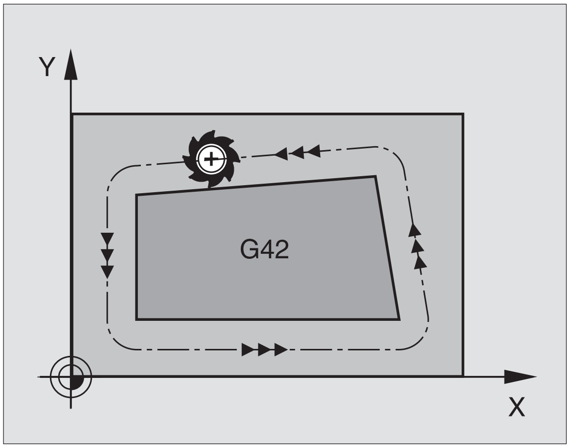


Contouring with radius compensation: G41 and G42

G42	The tool moves to the right of the programmed contour
G41	The tool moves to the left of the programmed contour

The tool center moves along the contour at a distance equal to the radius. "Right" or "left" are to be understood as based on the direction of tool movement along the workpiece contour. See figures at right.





> Between two program blocks with different radius compensations (**G42** and **G41**) you must program at least one traversing block in the working plane without radius compensation (that is, with **G40**).
>
> Radius compensation does not take effect until the end of the block in which it is first programmed.
>
> You can also activate the radius compensation for secondary axes in the working plane. Program the secondary axes as well in each following block, since otherwise the TNC will execute the radius compensation in the principal axis again.
>
> Whenever radius compensation is activated with **G42/G41** or canceled with G40, the TNC positions the tool perpendicular to the programmed starting or end position. Position the tool at a sufficient distance from the first or last contour point to prevent the possibility of damaging the contour.

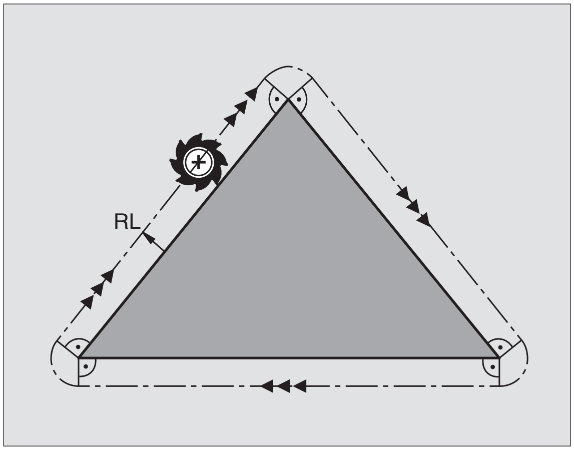
Entering radius compensation

Radius compensation is entered in a G01 block:

G **41**	To select tool movement to the left of the contour, select function G41, or
G **42**	To select tool movement to the right of the contour, select function G42, or
G **40**	To select tool movement without radius compensation or to cancel radius compensation, select function G40.
END	To terminate the block, press the END key.

Radius compensation: Machining corners

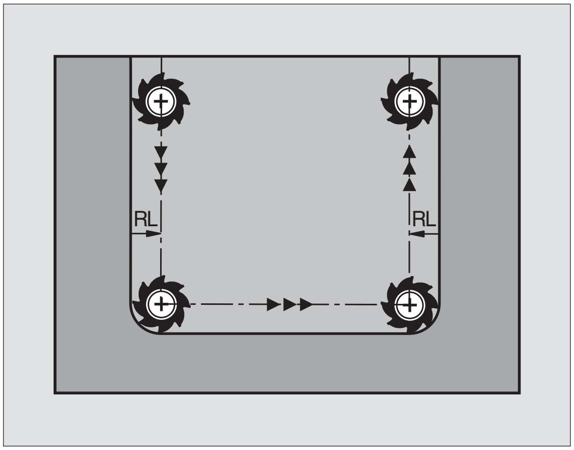
- Outside corners
  If you program radius compensation, the TNC moves the tool around outside corners either on a transitional arc or on a spline (selectable via MP7680). If necessary, the TNC reduces the feed rate at outside corners to reduce machine stress, for example at very great changes of direction.
- Inside corners
  The TNC calculates the intersection of the tool center paths at inside corners under radius compensation. From this point it then starts the next contour element. This prevents damage to the workpiece. The permissible tool radius, therefore, is limited by the geometry of the programmed contour.



> To prevent the tool from damaging the contour, be careful not to program the starting or end position for machining inside corners at a corner of the contour.

Machining corners without radius compensation

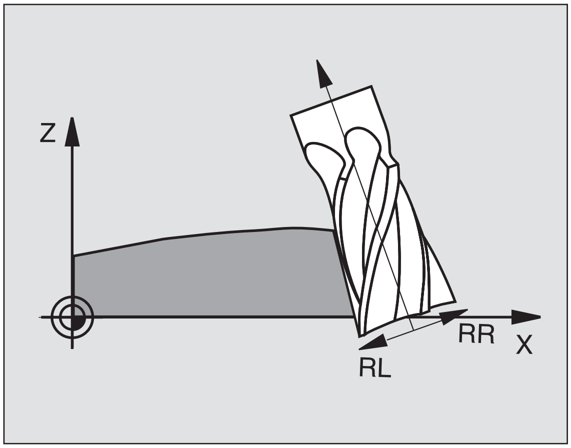
If you program the tool movement without radius compensation, you can change the tool path and feed rate at workpiece corners with the miscellaneous function **M90**, See "Smoothing corners: M90," page 153.



5.4 Peripheral Milling: 3-D Radius Compensation with Workpiece Orientation

Function

With peripheral milling, the TNC displaces the tool perpendicular to the direction of movement and perpendicular to the tool direction by the sum of the delta values **DR** (tool table and **T** block). Determine the compensation direction with radius compensation **G41/G42** (see figure at upper right, traverse direction Y+).



For the TNC to be able to reach the set tool orientation, you need to activate the function **M128** (see "Maintaining the position of the tool tip when positioning with tilted axes (TCPM*): M128 (not TNC 410)" on page 168) and subsequently the tool radius compensation. The TNC then positions the rotary axes automatically so that the tool can reach the orientation defined by the coordinates of the rotary axes with the active compensation.

The TNC is not able to automatically position the rotary axes on all machines. Refer to your machine manual.

Danger of collision

On machines whose rotary axes only allow limited traverse, sometimes automatic positioning can require the table to be rotated by 180°. In this case, make sure that the tool head does not collide with the workpiece or the clamps.

You can define the tool orientation in a G01 block as described below.

Example: Definition of the tool orientation with M128 and the coordinates of the rotary axes

`N10 G00 G90 X-20 Y+0 Z+0 B+0 C+0 *`	Pre-position
`N20 M128 *`	Activate M128
`N30 G01 G42 X+0 Y+0 Z+0 B+0 C+0 F1000 *`	Activate radius compensation
`N40 X+50 Y+0 Z+0 B-30 C+0 *`	Position rotary axis (tool orientation)

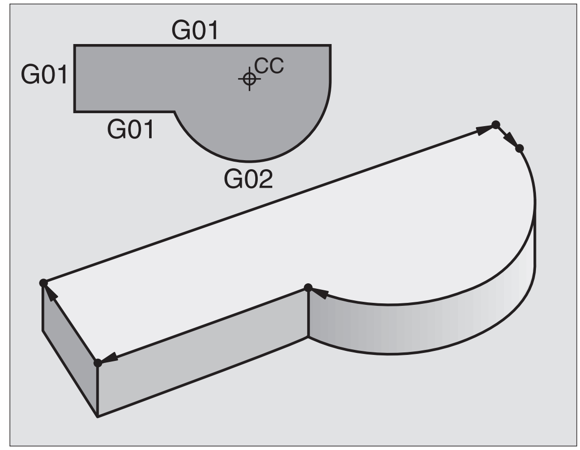
6

Programming: Programming Contours

6.1 Tool Movements

Path functions

A workpiece contour is usually composed of several contour elements such as straight lines and circular arcs. With the path functions, you can program the tool movements for **straight lines** and **circular arcs.**



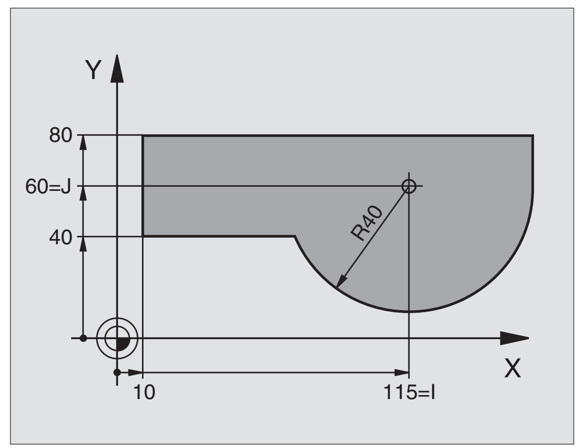
Miscellaneous functions M

With the miscellaneous functions of the TNC you can control:

- Program run, e.g., a program interruption
- Machine functions, such as switching spindle rotation and coolant supply on and off
- Contouring behavior of the tool

Subprograms and program section repeats

If a machining sequence occurs several times in a program, you can save time and reduce the chance of programming errors by entering the sequence once and then defining it as a subprogram or program section repeat. If you wish to execute a specific program section only under certain conditions, you also define this machining sequence as a subprogram. In addition, you can have a part program call a separate program for execution.



Programming with subprograms and program section repeats is described in Chapter 9.

Programming with Q parameters

Instead of programming numerical values in a part program, you enter markers called Q parameters. You assign the values to the Q parameters separately with the Q parameter functions. You can use the Q parameters for programming mathematical functions that control program execution or describe a contour.

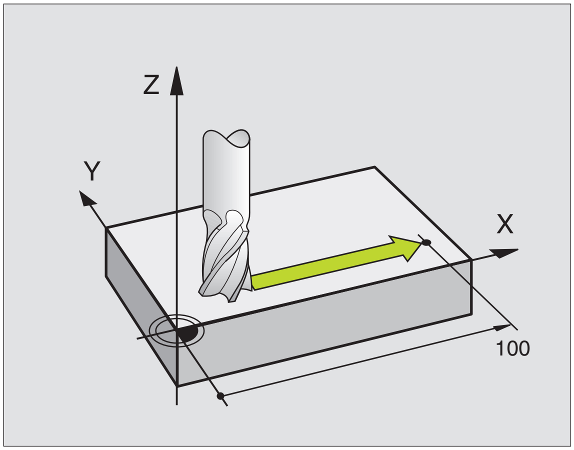
In addition, parametric programming enables you to measure with the 3-D touch probe during program run.

Programming with Q parameters is described in Chapter 10.

6.2 Fundamentals of Path Functions

Programming tool movements for workpiece machining

You create a part program by programming the path functions for the individual contour elements in sequence. You usually do this by entering **the coordinates of the end points of the contour elements** given in the production drawing. The TNC calculates the actual path of the tool from these coordinates, and from the tool data and radius compensation.



The TNC moves all axes programmed in a single block simultaneously.

Movement parallel to the machine axes

The program block contains only one coordinate. The TNC thus moves the tool parallel to the programmed axis.

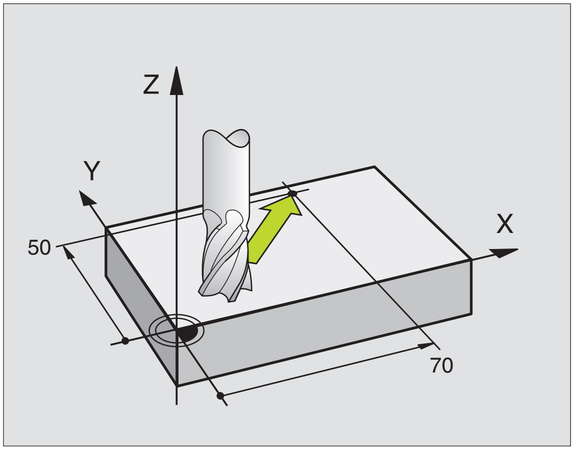
Depending on the individual machine tool, the part program is executed by movement of either the tool or the machine table on which the workpiece is clamped. Nevertheless, you always program path contours as if the tool moves and the workpiece remains stationary.

Example:

```
N50 G00 X+100 *
```

N50	Block number
G00	Path function "straight line at rapid traverse"
X+100	Coordinate of the end point

The tool retains the Y and Z coordinates and moves to the position X=100 (see figure at upper right).



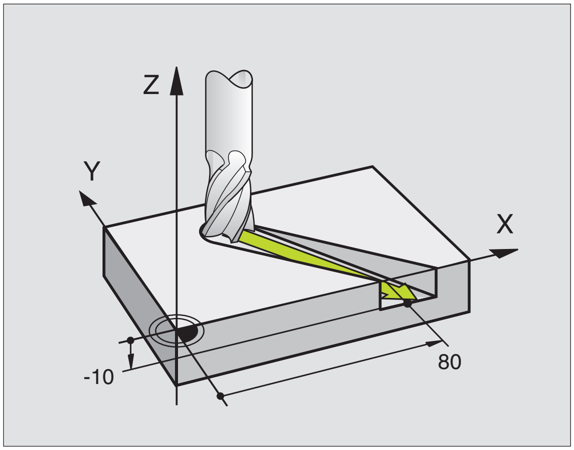
Movement in the main planes

The program block contains two coordinates. The TNC thus moves the tool in the programmed plane.

Example:

```
N50 G00 X+70 Y+50 *
```

The tool retains the Z coordinate and moves in the XY plane to the position X=70, Y=50 (see figure at center right).



Three-dimensional movement

The program block contains three coordinates. The TNC thus moves the tool in space to the programmed position.

Example:

```
N50 G01 X+80 Y+0 Z-10 *
```

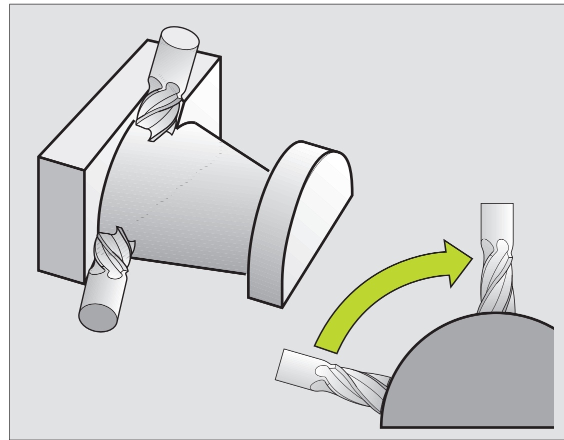
Entering more than three coordinates (not TNC 410)

The TNC can control up to 5 axes simultaneously. Machining with 5 axes, for example, moves 3 linear and 2 rotary axes simultaneously.

Such programs are too complex to program at the machine, however, and are usually created with a CAD system.

Example:

```
N G01 G40 X+20 Y+10 Z+2 A+15 C+6 F100 M3 *
```

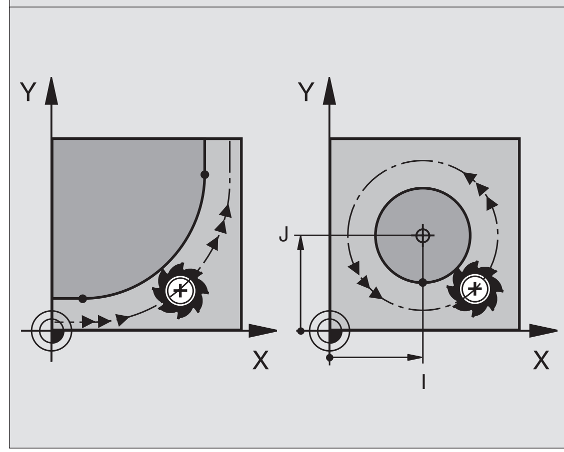


> The TNC graphics cannot simulate movements in more than three axes.

Circles and circular arcs

The TNC moves two axes simultaneously in a circular path relative to the workpiece. You can define a circular movement by entering a circle center.

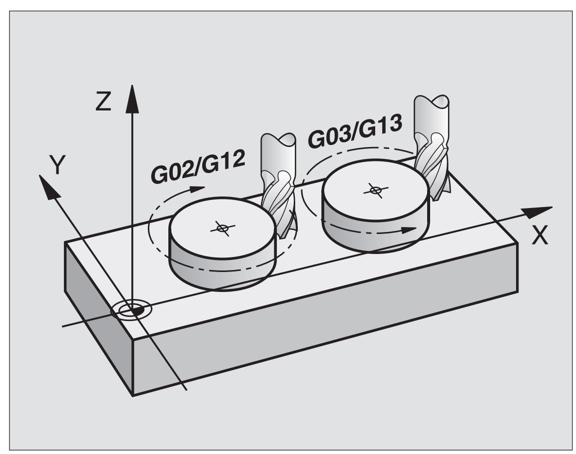
When you program a circle, the TNC assigns it to one of the main planes. This plane is defined automatically when you set the spindle axis during a tool call:



Tool axis	Main plane	Circle center
Z (G17)	**XY**, also UV, XV, UY	**I, J**
Y (G18)	**ZX**, also WU, ZU, WX	**K, I**
X (G19)	**YZ**, also VW, YW, VZ	**J, K**

> You can program circles that do not lie parallel to a main plane by using the function for tilting the working plane (see "WORKING PLANE (Cycle G80, not TNC 410)," page 304) or Q parameters (see "Principle and Overview," page 330).

Direction of rotation for circular movements



When a circular path has no tangential transition to another contour element, enter the direction of rotation with the following functions:

- Clockwise direction of rotation: G02/G12
- Counterclockwise direction of rotation: G03/G13

Radius compensation

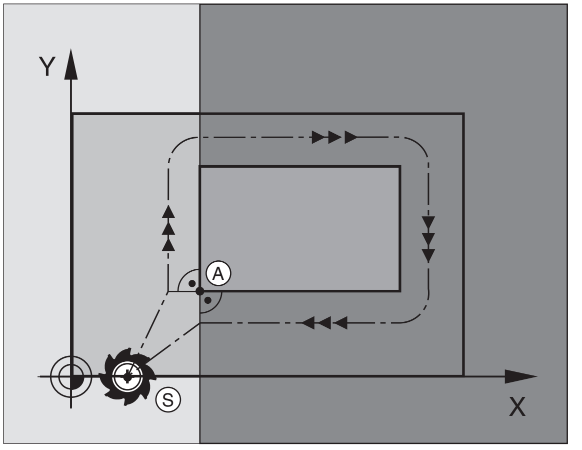
The radius compensation must be in the block in which you move to the first contour element. You cannot begin radius compensation in a circle block. It must be activated beforehand in a straight-line block (see "Path Contours—Cartesian Coordinates," page 126).

Pre-positioning

Before running a part program, always pre-position the tool to prevent the possibility of damaging it or the workpiece.

6.3 Contour Approach and Departure

Starting point and end point



The tool approaches the first contour point from the starting point. The starting point must be:

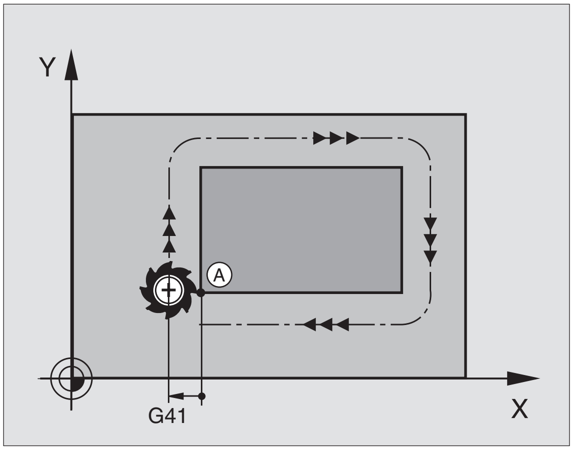
- Programmed without radius compensation
- Approachable without danger of collision
- Close to the first contour point

Example

Figure at upper right: If you set the starting point in the dark gray area, the contour will be damaged when the first contour element is approached.

First contour point

You need to program a radius compensation for the tool movement to the first contour point.

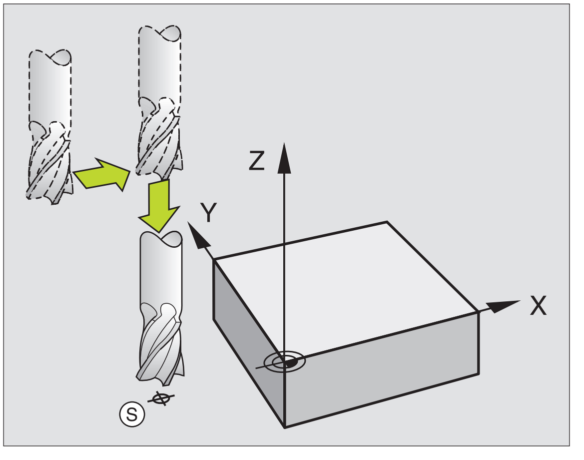


Approaching the starting point in the spindle axis

When the starting point is approached, the tool must be moved to the working depth in the spindle axis. If danger of collision exists, approach the starting point in the spindle axis separately.

Example NC blocks

```
N30 G00 G40 X+20 Y+30 *
N40 Z-10 *
```



End point

The end point should be selected so that it is:

- Approachable without danger of collision
- Near to the last contour point
- In order to make sure the contour will not be damaged, the optimal ending point should lie on the extended tool path for machining the last contour element.

Example

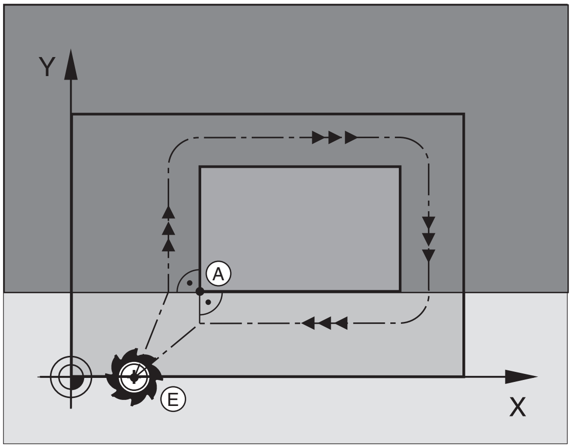
Figure at upper right: If you set the ending point in the dark gray area, the contour will be damaged when the end point is approached.

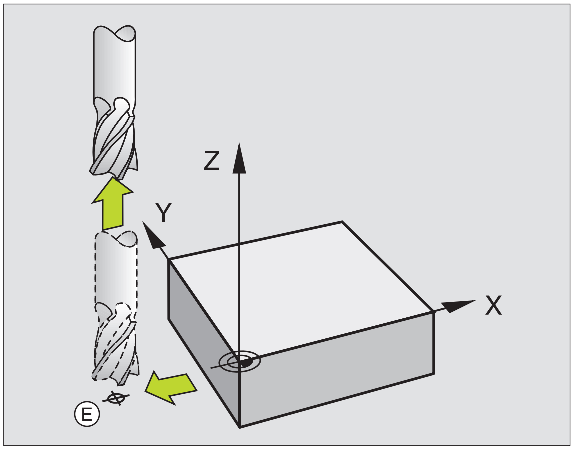
Depart the end point in the spindle axis:

Program the departure from the end point in the spindle axis separately. See figure at center right.

Example NC blocks

```
N50 G00 G40 X+60 Y+70 *
N60 Z+250 *
```





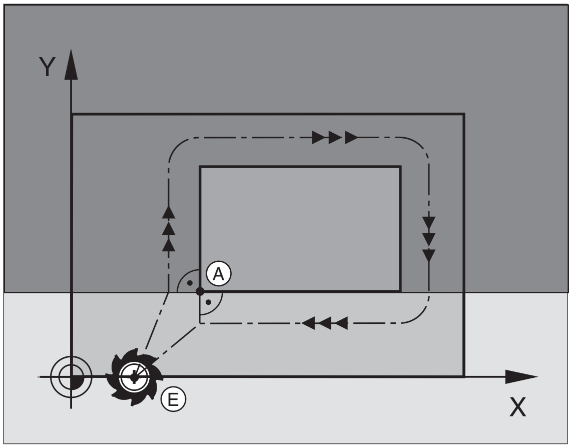
Common starting and end points

Do not program any radius compensation if the starting point and end point are the same.

In order to make sure the contour will not be damaged, the optimal starting point should lie between the extended tool paths for machining the first and last contour elements.

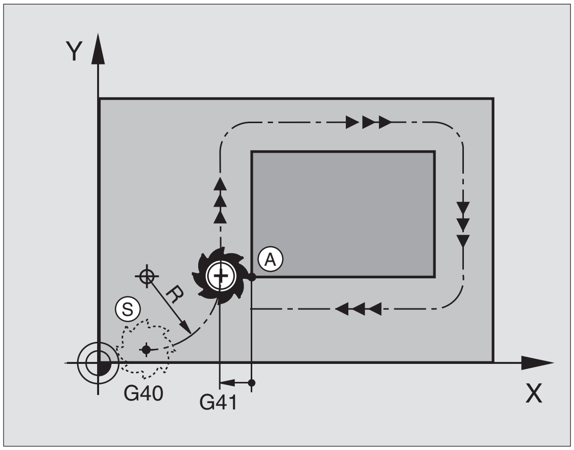
Example

Figure at upper right: If you set the starting point in the dark gray area, the contour will be damaged when the first contour element is approached.



Tangential approach and departure

With **G26** (figure at center right), you can program a tangential approach to the workpiece, and with **G27** (figure at lower right) a tangential departure. In this way you can avoid dwell marks.



Starting point and end point

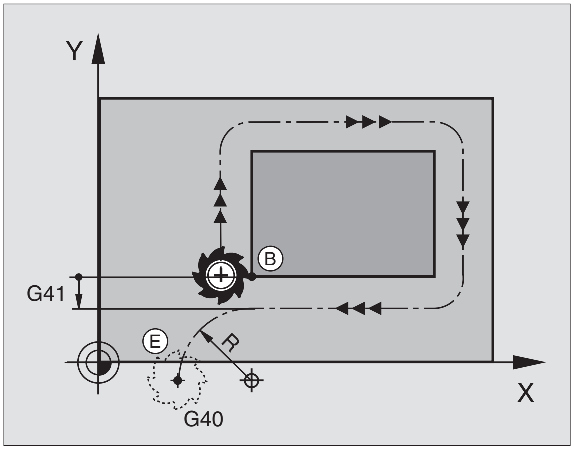
The starting point and the end point lie outside the workpiece, close to the first and last contour points. They are to be programmed without radius compensation.

Approach

- **G26** is entered after the block in which the first contour element is programmed: This will be the first block with radius compensation **G41/G42.**

Departure

- **G27** after the block in which the last contour element is programmed: This will be the last block with radius compensation **G41/G42.**



> The radius for **G26** and **G27** must be selected so that the TNC can execute the circular path between the starting point and the first contour point, as well as the last contour point and the end point.

Example NC blocks

`N50 G00 G40 G90 X-30 Y+50 *`	Starting position
`N60 G01 G41 X+0 Y+50 F350 *`	First contour point
`N70 G26 R5 *`	Tangential approach with radius R = 5 mm
`. . .`	
`PROGRAM CONTOUR BLOCKS`	
`. . .`	Last contour point
`N210 G27 R5 *`	Tangential departure with radius R = 5 mm
`N220 G00 G40 X-30 Y+50 *`	End point

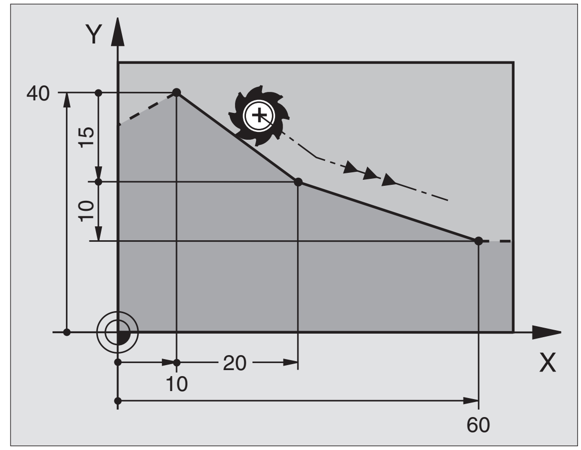
6.4 Path Contours—Cartesian Coordinates

Overview of path functions

Tool movement	Function	Required input
Straight line at feed rate Straight line at rapid traverse	**G00** **G01**	Coordinates of the end points of the straight line
Chamfer between two straight lines	**G24**	Length of chamfer **R**
–	**I**, **J**, **K**	Coordinates of the circle center
Circular path in clockwise direction Circular path in counterclockwise direction	**G02** **G03**	Coordinates of the arc end point in connection with **I**, **J**, **K** or additional circular radius **R**
Circular path corresponding to active direction of rotation	**G05**	Coordinates of the arc end point and circular radius **R**
Circular arc with tangential connection to the preceding contour element	**G06**	Coordinates of the arc end point
Circular arc with tangential connection to the preceding and subsequent contour elements	**G25**	Rounding-off radius **R**

Straight line at rapid traverse G00
Straight line with feed rate G01 F. . .

The TNC moves the tool in a straight line from its current position to the straight-line end point. The starting point is the end point of the preceding block.



Programming

G 1

- **`Coordinates`** of the end point of the straight line

Further entries, if necessary:

- **`Radius compensation G40/G41/G42`**
- **`Feed rate F`**
- **`Miscellaneous function M`**

Example NC blocks

```
N70 G01 G41 X+10 Y+40 F200 M3 *
N80 G91 X+20 Y-15  *
N90 G90 X+60 G91 Y-10 *
```

Actual position capture

You can capture any desired axis position by pressing the ACTUAL-POSITION-CAPTURE key:

- In the Manual Operation mode, move the tool to the position you wish to capture.
- Switch the screen display to Programming and Editing.
- Select the program block into which you want to take over an axis position.

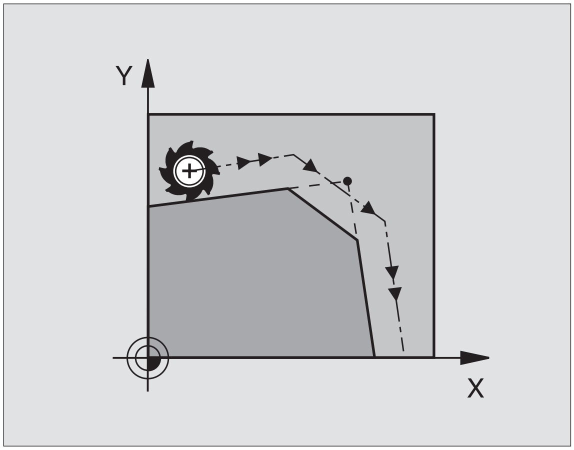
X

- Select the axis whose position you want to capture.
- Press the ACTUAL-POSITION-CAPTURE key: The TNC captures the coordinates of the actual position in the selected axis.

Inserting a chamfer CHF between two straight lines

The chamfer enables you to cut off corners at the intersection of two straight lines.

- The blocks before and after the **G24** block must be in the same working plane.
- The radius compensation before and after the **G24** block must be the same.
- An inside chamfer must be large enough to accommodate the current tool.



Programming

G 24

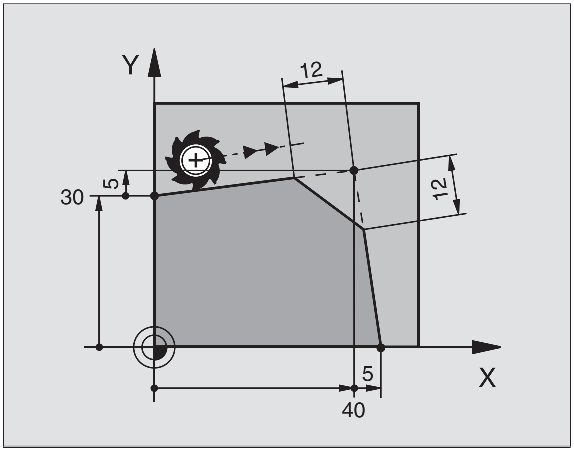
▶ **Chamfer side length:** Length of the chamfer

Further entries, if necessary:

▶ **Feed rate F** (only effective in **G24** block)

Example NC blocks

```
N70 G01 G41 X+0 Y+30 F300 M3 *
N80 X+40 G91 Y+5 *
N90 G24 R12 F250 *
N100 G91 X+5 G90 Y+0 *
```



You cannot start a contour with a **G24** block.

A chamfer is possible only in the working plane.

The corner point is cut off by the chamfer and is not part of the contour.

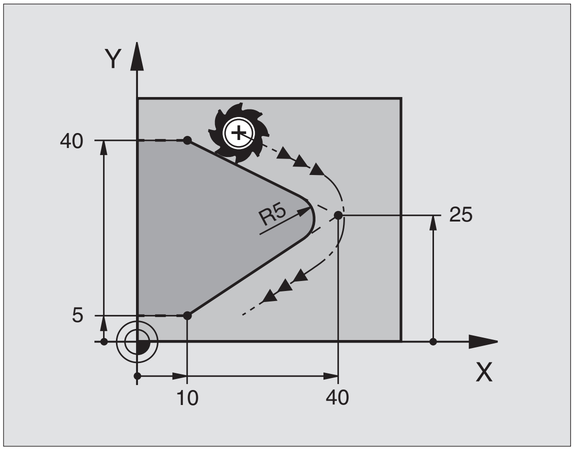
A feed rate programmed in the **G24** block is effective only in that block. After the **G24** block, the previous feed rate becomes effective again.

Rounding corners G25

The G25 function is used for rounding off corners.

The tool moves on an arc that is tangentially connected to both the preceding and subsequent contour elements.

The rounding arc must be large enough to accommodate the tool.



Programming

G 25

- **Rounding-off radius:** Enter the radius

Further entries, if necessary:
- **Feed rate F** (only effective in **G25** block)

Example NC blocks

```
N50 G01 G41 X+10 Y+40 F300 M3 *
N60 X+40 Y+25 *
N70 G25 R5 F100 *
N80 X+10 Y+5 *
```

In the preceding and subsequent contour elements, both coordinates must lie in the plane of the rounding arc. If you machine the contour without tool-radius compensation, you must program both coordinates in the working plane.

The corner point is cut off by the rounding arc and is not part of the contour.

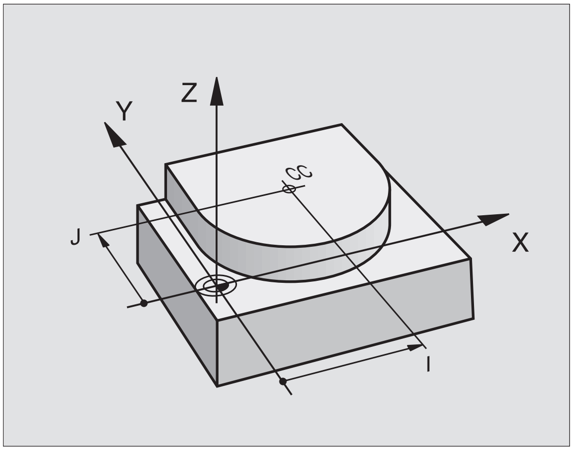
A feed rate programmed in the **G25** block is effective only in that block. After the **G25** block, the previous feed rate becomes effective again.

You can also use a **G25** block for a tangential contour approach, see "Tangential approach and departure," page 124.

Circle center I, J

You can define a circle center for circles that are programmed with the functions G02, G03 or G05. This is done in the following ways:

- Entering the Cartesian coordinates of the circle center, or
- Using the circle center defined in an earlier block, or
- Capturing the coordinates with the ACTUAL-POSITION-CAPTURE key.



Programming

I J

▶ Enter the coordinates for the circle center, or if you want to use the last programmed position, enter G29.

Example NC blocks

```
N50 I+25 J+25 *
```

or

```
N10 G00 G40 X+25 Y+25 *
N20 G29 *
```

The program blocks N10 and N20 do not refer to the illustration.

Duration of effect

The circle center definition remains in effect until a new circle center is programmed. You can also define a circle center for the secondary axes U, V and W.

Entering incremental values for the circle center I, J

If you enter the circle center with incremental coordinates, you have programmed it relative to the last programmed position of the tool.

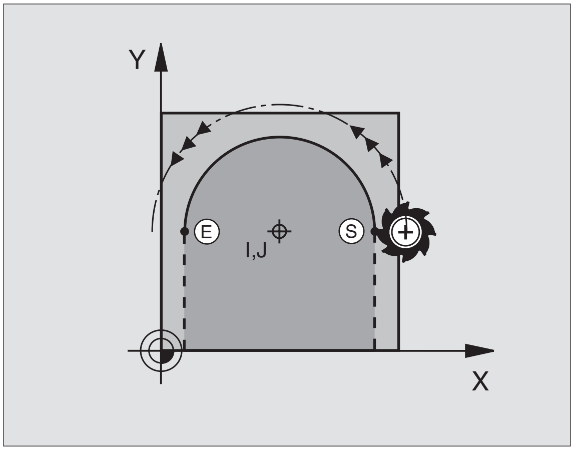
The only effect of **I** and **J** is to define a position as a circle center—the tool does not move to the position.

The circle center is also the pole for polar coordinates.

If you wish to define the pole in parallel axes, first press the **I** (**J**) key on the ASCII keyboard, and then the orange axis key for the corresponding parallel axis.

Circular path G02/G03/G05 around circle center I, J

Before programming a circular arc, you must first enter the circle center **I**, **J.** The last programmed tool position will be the starting point of the arc.



Direction

- In clockwise direction: **G02**
- In counterclockwise direction: **G03**
- Without programmed direction: **G05.** The TNC traverses the circular arc with the last programmed direction of rotation.

Programming

▶ Move the tool to the circle starting point.

I J ▶ Enter the coordinates of the circle center.

G 3 ▶ Enter the coordinates of the arc end point.

Further entries, if necessary:

▶ Feed rate F

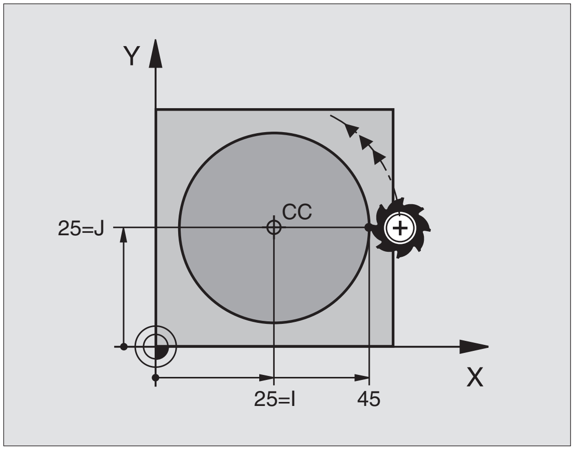
▶ Miscellaneous function M

Example NC blocks

```
N50 I+25 J+25 *
N60 G01 G42 X+45 Y+25 F200 M3 *
N70 G03 X+45 Y+25 *
```

Full circle

Enter the same point you used as the starting point for the end point in a C block.

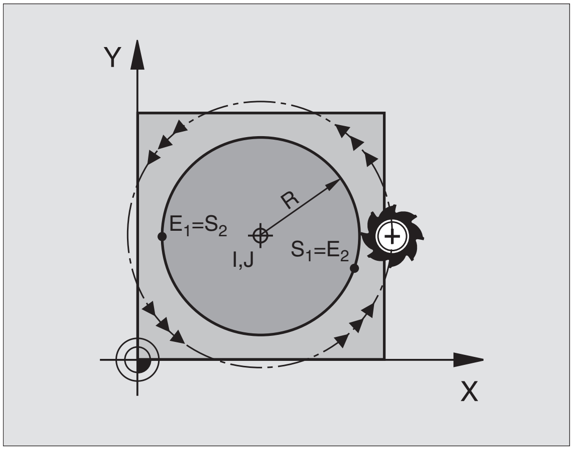


The starting and end points of the arc must lie on the circle.

Input tolerance: up to 0.016 mm (selected with MP7431, not for TNC 410)

Circular path G02/G03/G05 with defined radius

The tool moves on a circular path with the radius R.



Direction

- In clockwise direction: **G02**
- In counterclockwise direction: **G03**
- Without programmed direction: **G05.** The TNC traverses the circular arc with the last programmed direction of rotation.

Programming

G 3

- Enter the coordinates of the arc end point.
- Radius R
  Note: The algebraic sign determines the size of the arc!

Further entries, if necessary:
- Feed rate F
- Miscellaneous function M

Full circle

For a full circle, program two CR blocks in succession:

The end point of the first semicircle is the starting point of the second. The end point of the second semicircle is the starting point of the first.

Central angle CCA and arc radius R

The starting and end points on the contour can be connected with four arcs of the same radius:

Smaller arc: CCA<180°
Enter the radius with a positive sign R>0

Larger arc: CCA>180°
Enter the radius with a negative sign R<0

The direction of rotation determines whether the arc is curving outward (convex) or curving inward (concave):

Convex: Direction of rotation **G02** (with radius compensation **G41**)

Concave: Direction of rotation **G03** (with radius compensation **G41**)

Example NC blocks

```
N100 G01 G41 X+40 Y+40 F200 M3 *
N110 G02 X+70 Y+40 R+20 * (ARC 1)
```

or

```
N110 G03 X+70 Y+40 R+20 * (ARC 2)
```

or

```
N110 G02 X+70 Y+40 R-20 * (ARC 3)
```

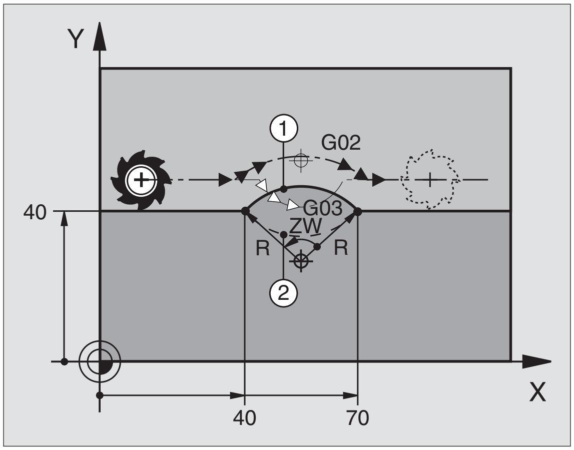
or

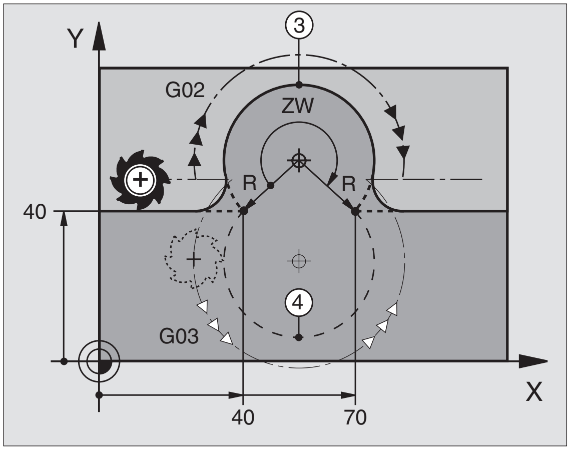
```
N110 G03 X+70 Y+40 R-20 * (ARC 4)
```

The distance from the starting and end points of the arc diameter cannot be greater than the diameter of the arc.

The maximum radius is 99.9999 m.

You can also enter rotary axes A, B and C.



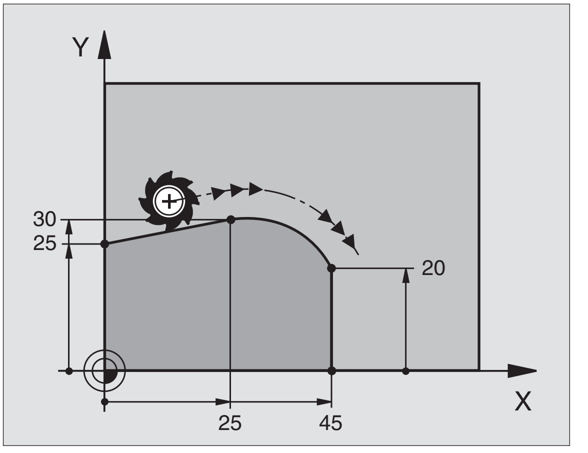


Circular path G06 with tangential approach

The tool moves on an arc that starts at a tangent with the previously programmed contour element.

A transition between two contour elements is called tangential when there is no kink or corner at the intersection between the two contours—the transition is smooth.

The contour element to which the tangential arc connects must be programmed immediately before the **G06** block. This requires at least two positioning blocks.



Programming

G 6

- Enter the coordinates of the arc end point.

Further entries, if necessary:

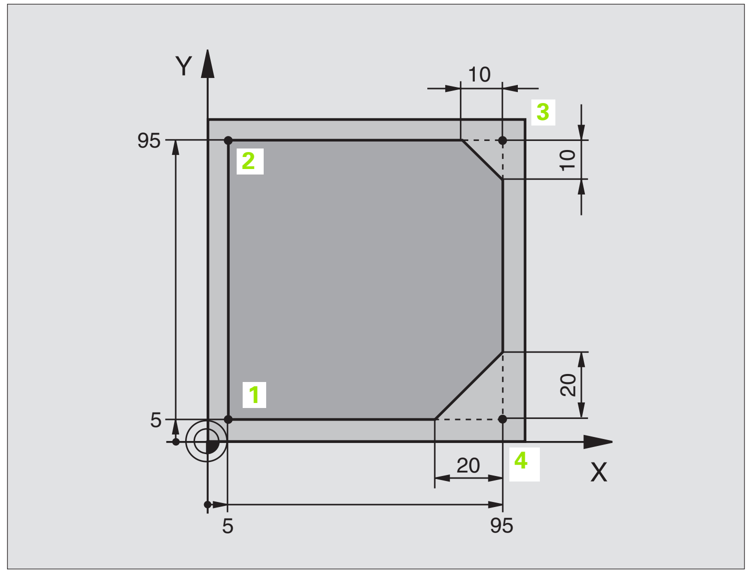
- Feed rate F
- Miscellaneous function M

Example NC blocks

```
N70 G01 G41 X+0 Y+25 F300 M3 *
N80 X+25 Y+30 *
N90 G06 X+45 Y+20 *
G01 Y+0 *
```

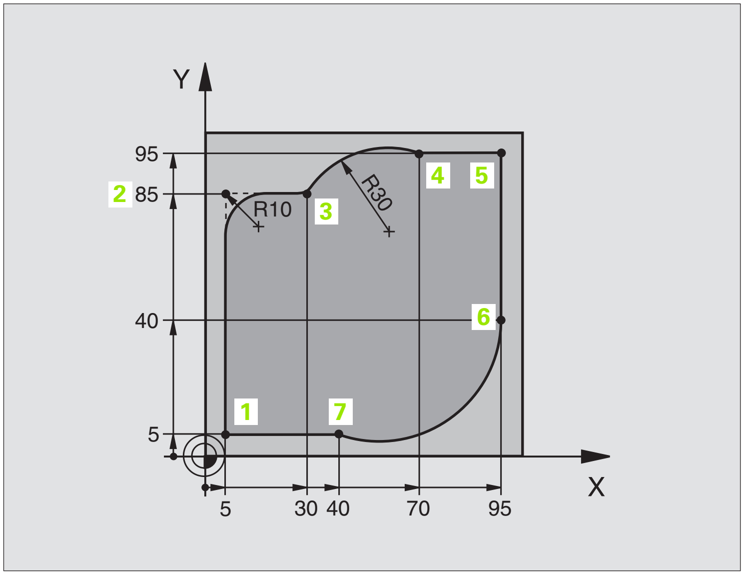
A tangential arc is a two-dimensional operation: the coordinates in the **G06** block and in the contour element preceding it must be in the same plane of the arc.

Example: Linear movements and chamfers with Cartesian coordinates



`%LINEAR G71 *`	
`N10 G30 G17 X+0 Y+0 Z-20 *`	Define blank form for graphic workpiece simulation
`N20 G31 G90 X+100 Y+100 Z+0 *`	
`N30 G99 T1 L+0 R+10 *`	Define tool in the program
`N40 T1 G17 S4000 *`	Call tool in the spindle axis and with the spindle speed S
`N50 G00 G40 G90 Z+250 *`	Retract tool in the spindle axis at rapid traverse
`N60 X-10 Y-10 *`	Pre-position the tool
`N70 G01 Z-5 F1000 M3 *`	Move to working depth at feed rate F = 1000 mm/min
`N80 G01 G41 X+5 Y+5 F300 *`	Approach the contour at point 1, activate radius compensation G41
`N90 G26 R5 F150 *`	Tangential approach
`N100 Y+95 *`	Move to point 2
`N110 X+95 *`	Point 3: first straight line for corner 3
`N120 G24 R10 *`	Program chamfer with length 10 mm
`N130 Y+5 *`	Point 4: 2nd straight line for corner 3, 1st straight line for corner 4
`N140 G24 R20 *`	Program chamfer with length 20 mm
`N150 X+5 *`	Move to last contour point 1, second straight line for corner 4
`N160 G27 R5 F500 *`	Tangential departure
`N170 G40 X-20 Y-20 F1000 *`	Retract tool in the working plane, cancel radius compensation
`N180 G00 Z+250 M2 *`	Retract in the tool axis, end program
`N999999 %LINEAR G71 *`	

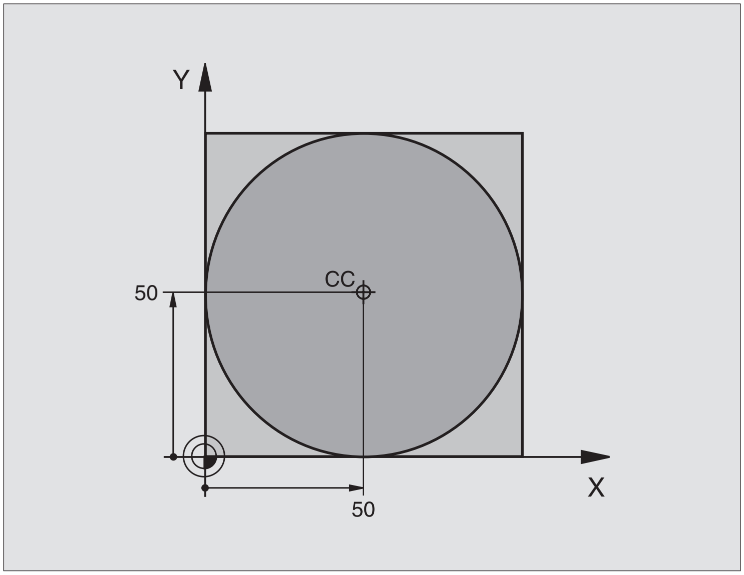
Example: Circular movements with Cartesian coordinates



%CIRCULAR G71 *	
N10 G30 G17 X+0 Y+0 Z-20 *	Define blank form for graphic workpiece simulation
N20 G31 G90 X+100 Y+100 Z+0 *	
N30 G99 T1 L+0 R+10 *	Define tool in the program
N40 T1 G17 S4000 *	Call tool in the spindle axis and with the spindle speed S
N50 G00 G40 G90 Z+250 *	Retract tool in the spindle axis at rapid traverse
N60 X-10 Y-10 *	Pre-position the tool
N70 G01 Z-5 F1000 M3 *	Move to working depth at feed rate F = 1000 mm/min
N80 G01 G41 X+5 Y+5 F300 *	Approach the contour at point 1, activate radius compensation G41
N90 G26 R5 F150 *	Tangential approach
N100 Y+85 *	Point 2: first straight line for corner 2
N110 G25 R10 *	Insert radius with R = 10 mm, feed rate: 150 mm/min
N120 X+30 *	Move to point 3: Starting point of the arc
N130 G02 X+70 Y+95 R+30 *	Move to point 4: end point of the arc with G02, radius 30 mm
N140 G01 X+95 *	Move to point 5
N150 Y+40 *	Move to point 6
N160 G06 X+40 Y+5 *	Move to point 7: End point of the arc, radius with tangential
	connection to point 6, TNC automatically calculates the radius

`N170 G01 X+5 *`	Move to last contour point 1
`N180 G27 R5 F500 *`	Depart the contour on a circular arc with tangential connection
`N190 G40 X-20 Y-20 F1000 *`	Retract tool in the working plane, cancel radius compensation
`N200 G00 Z+250 M2 *`	Retract tool in the tool axis, end of program
`N999999 %CIRCULAR G71 *`	

Example: Full circle with Cartesian coordinates



%C-CC G71 *	
N10 G30 G17 X+0 Y+0 Z-20 *	Define the workpiece blank
N20 G31 G90 X+100 Y+100 Z+0 *	
N30 G99 T1 L+0 R+12.5 *	Define the tool
N40 T1 G17 S3150 *	Tool call
N50 G00 G40 G90 Z+250 *	Retract the tool
N60 I+50 J+50 *	Define the circle center
N70 X-40 Y+50 *	Pre-position the tool
N80 G01 Z-5 F1000 M3 *	Move to working depth
N90 G41 X+0 Y+50 F300 *	Approach starting point, radius compensation G41
N100 G26 R5 F150 *	Tangential approach
N110 G02 X+0 *	Move to the circle end point (= circle starting point)
N120 G27 R5 F500 *	Tangential departure
N130 G01 G40 X-40 Y-50 F1000 *	Retract tool in the working plane, cancel radius compensation
N140 G00 Z+250 M2 *	Retract tool in the tool axis, end of program
N999999 %C-CC G71 *	

6.5 Path Contours—Polar Coordinates

Overview of path functions with polar coordinates

With polar coordinate you can define a position in terms of its angle H and its distance R relative to a previously defined pole **I, J** (see "Definition of pole and angle reference axis," page 40).

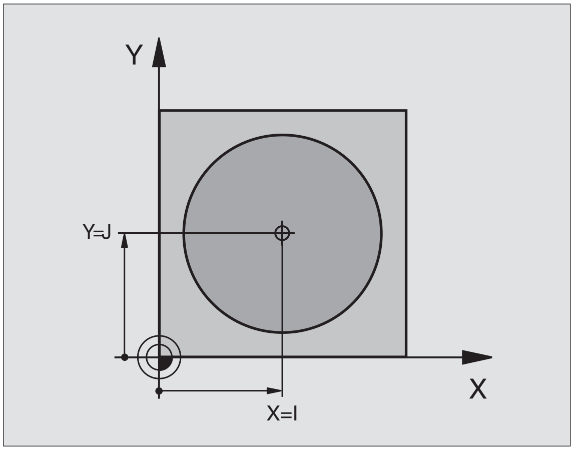
Polar coordinates are useful with:

- Positions on circular arcs
- Workpiece drawing dimensions in degrees, e.g. bolt hole circles

Tool movement	Function	Required input
Straight line at feed rate Straight line at rapid traverse	**G10** **G11**	Polar radius, polar angle of the straight-line end point
Circular path in clockwise direction Circular path in counterclockwise direction	**G12** **G13**	Polar angle of the circle end point
Circular path corresponding to active direction of rotation	**G15**	Polar angle of the circle end point
Circular arc with tangential connection to the preceding contour element	**G16**	Polar radius, polar angle of the arc end point

Zero point for polar coordinates: pole I, J

You can set the pole **I**, **J** at any point in the machining program, before indicating points in polar coordinates. Set the pole in the same way as you would program the circle center.



Programming

I J

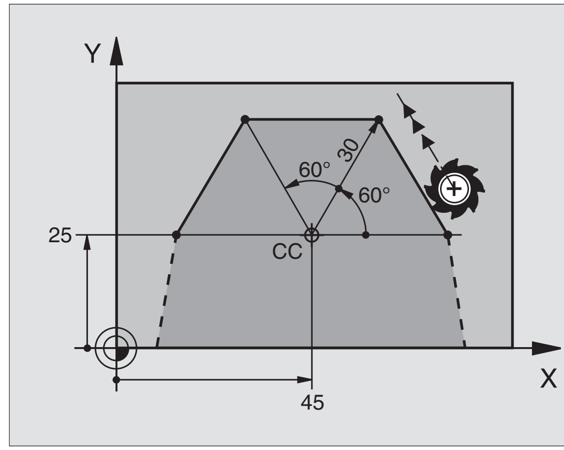
▶ Enter Cartesian coordinates for the pole, or if you want to use the last programmed position, enter **G29.** Before programming polar coordinates, define the pole. You can only define the pole in Cartesian coordinates. The pole remains in effect until you define a new pole.

Example NC blocks

```
N120 I+45 J+45 *
```

Straight line at rapid traverse G10
Straight line with feed rate G11 F . . .

The tool moves in a straight line from its current position to the straight-line end point. The starting point is the end point of the preceding block.



Programming

G 11

- Polar coordinates radius **R:** Enter distance from the straight line end point to the pole **I**, **J**
- Polar-coordinates angle **H:** Angular position of the straight-line end point between -360° and +360°

The sign of **H** depends on the angle reference axis:

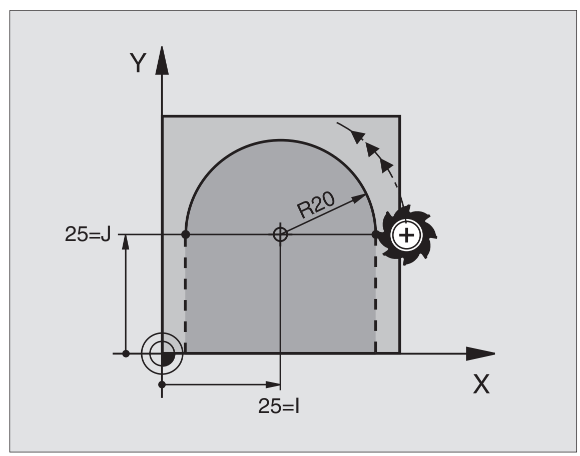
- Angle from angle reference axis to **R** is counterclockwise: **H** >0
- Angle from angle reference axis to **R** is clockwise: **H** <0

Example NC blocks

```
N120 I+45 J+45 *
N130 G11 G42 R+30 H+0 F300 M3 *
N140 H+60 *
N150 G91 H+60 *
N160 G90 H+180 *
```

Circular path G12/G13/G15 around pole I, J

The polar coordinate radius **R** is also the radius of the arc. It is defined by the distance from the starting point to the pole **I**, **J**. The last programmed tool position before the **G12**, **G13** or **G15** block is the starting point of the arc.



Direction

- In clockwise direction: **G12**
- In counterclockwise direction: **G13**
- Without programmed direction: **G15.** The TNC traverses the circular arc with the last programmed direction of rotation.

Programming

G 13

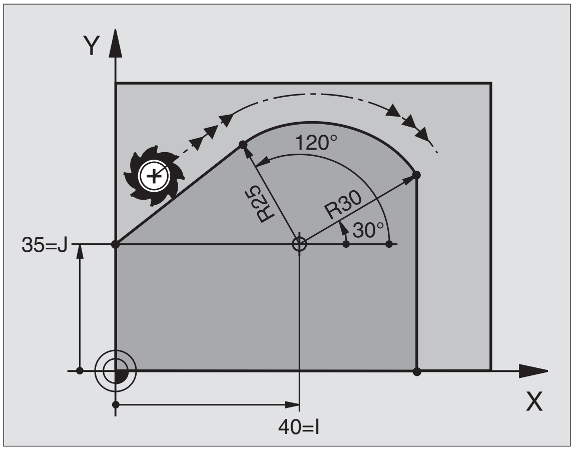
- Polar-coordinates angle **H:** Angular position of the arc end point between -5400° and +5400°

Example NC blocks

```
N180 I+25 J+25 *
N190 G11 G42 R+20 H+0 F250 M3 *
N200 G13 H+180 *
```

Circular arc with tangential connection

The tool moves on a circular path, starting tangentially from a preceding contour element.



Programming

G 16

- Polar coordinates radius **R:** Distance from the arc end point to the pole **I**, **J**
- Polar coordinates angle **H:** Angular position of the arc end point

Example NC blocks

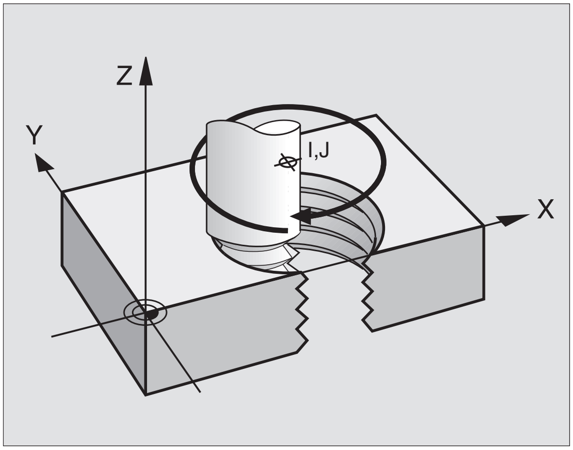
```
N120 I+40 J+35 *
N130 G01 G42 X+0 Y+35 F250 M3 *
N140 G11 R+25 H+120 *
N150 G16 R+30 H+30 *
N160 G01 Y+0 *
```

The pole is **not** the center of the contour arc!

Helical interpolation

A helix is a combination of a circular movement in a main plane and a line movement perpendicular to this plane.

A helix is programmed only in polar coordinates.



Application

- Large-diameter internal and external threads
- Lubrication grooves

Calculating the helix

To program a helix, you must enter the total angle through which the tool is to move on the helix in incremental dimensions, and the total height of the helix.

For calculating a helix that is to be cut in a upward direction, you need the following data:

Thread revolutions *n*	Thread revolutions + thread overrun at the start and end of the thread
Total height *h*	Thread pitch P times thread revolutions *n*
Incremental total angle H	Number of revolutions times 360° + angle for beginning of thread + angle for thread overrun
Starting coordinate Z	Pitch P times (thread revolutions + thread overrun at start of thread)

Shape of the helix

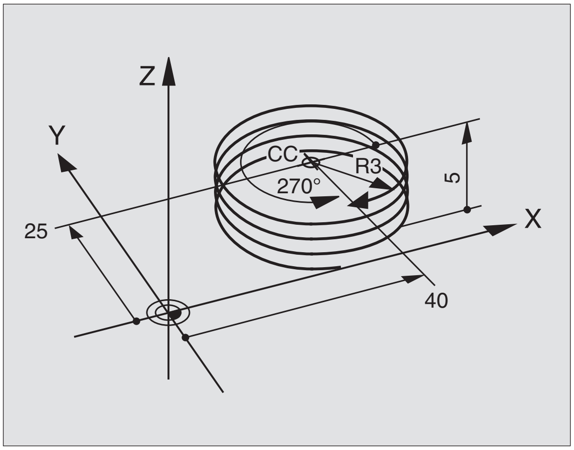
The table below illustrates in which way the shape of the helix is determined by the work direction, direction of rotation and radius compensation.

Internal thread	Work direction	Direction	Radius comp.
Right-handed	Z+	**G13**	**G41**
Left-handed	Z+	**G12**	**G42**
Right-handed	Z–	**G12**	**G42**
Left-handed	Z–	**G13**	**G41**
External thread			
Right-handed	Z+	**G13**	**G42**
Left-handed	Z+	**G12**	**G41**
Right-handed	Z–	**G12**	**G41**
Left-handed	Z–	**G13**	**G42**

Programming a helix

Always enter the same algebraic sign for the direction of rotation and the incremental total angle **G91 H.** The tool may otherwise move in a wrong path and damage the contour.

For the total angle **G91 H,** you can enter a value from –5400° to +5400°. If the thread has more than 15 revolutions, program the helix in a program section repeat (see "Program Section Repeats," page 318)



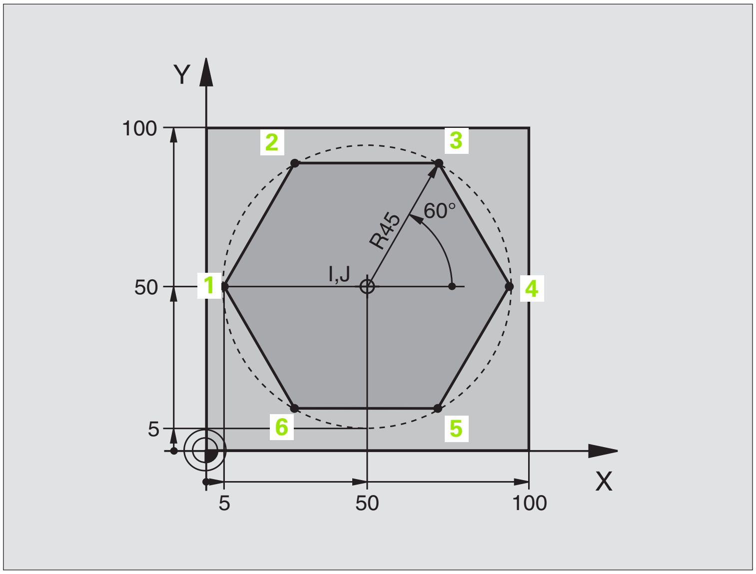
G 12

- Polar coordinates angle H: Enter the total angle of tool traverse along the helix in incremental dimensions. **After entering the angle, specify the tool axis with an axis selection key.**
- Enter the coordinate for the height of the helix in incremental dimensions.
- Enter the radius compensation **G41/G42** according to the table above.

Example NC blocks: Thread M6 x 1 mm with 5 revolutions

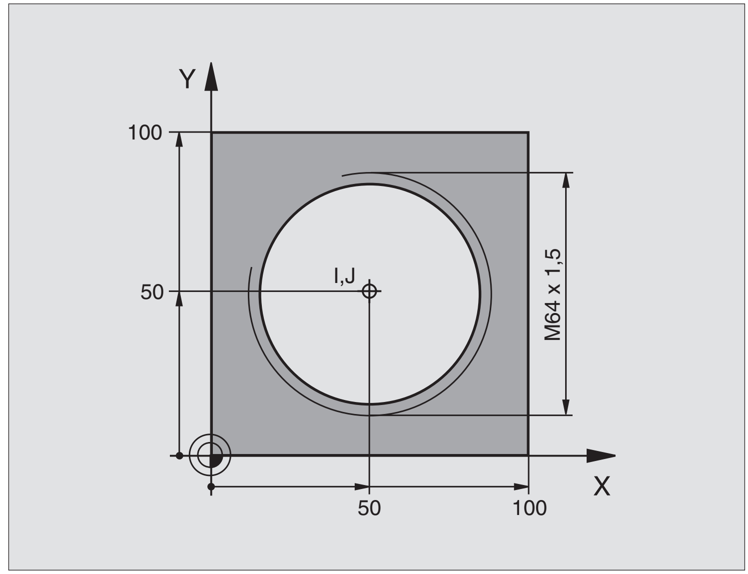
```
N120 I+40 J+25 *
N130 G01 Z+0 F100 M3 *
N140 G11 G41 R+3 H+270 *
N150 G12 G91 H-1800 Z+5 *
```

Example: Linear movement with polar coordinates



%LINEARPO G71 *	
N10 G30 G17 X+0 Y+0 Z-20 *	Define the workpiece blank
N20 G31 G90 X+100 Y+100 Z+0 *	
N30 G99 T1 L+0 R+7.5 *	Define the tool
N40 T1 G17 S4000 *	Tool call
N50 G00 G40 G90 Z+250 *	Define the datum for polar coordinates
N60 I+50 J+50 *	Retract the tool
N70 G10 R+60 H+180 *	Pre-position the tool
N80 G01 Z-5 F1000 M3 *	Move to working depth
N90 G11 G41 R+45 H+180 F250 *	Approach the contour at point 1
N110 G26 R5 *	Approach the contour at point 1
N120 H+120 *	Move to point 2
N130 H+60 *	Move to point 3
N140 H+0 *	Move to point 4
N150 H-60 *	Move to point 5
N160 H-120 *	Move to point 6
N170 H+180 *	Move to point 1
N180 G27 R5 F500 *	Tangential departure
N190 G40 R+60 H+180 F1000 *	Retract tool in the working plane, cancel radius compensation
N200 G00 Z+250 M2 *	Retract in the spindle axis, end of program
N999999 %LINEARPO G71 *	

Example: Helix



`%HELIX G71 *`	
`N10 G30 G17 X+0 Y+0 Z-20 *`	Define the workpiece blank
`N20 G31 G90 X+100 Y+100 Z+0 *`	
`N30 G99 T1 L+0 R+5 *`	Define the tool
`N40 T1 G17 S1400 *`	Tool call
`N50 G00 G40 G90 Z+250 *`	Retract the tool
`N60 X+50 Y+50 *`	Pre-position the tool
`N70 G29 *`	Transfer the last programmed position as the pole
`N80 G01 Z-12.75 F1000 M3 *`	Move to working depth
`N90 G11 G41 R+32 H+180 F250 *`	Approach first contour point
`N100 G26 R2 *`	Connection
`N110 G13 G91 H+3240 Z+13.5 F200 *`	Helical interpolation
`N120 G27 R2 F500 *`	Tangential departure
`N170 G01 G40 G90 X+50 Y+50 F1000 *`	Retract in the tool axis, end program
`N180 G00 Z+250 M2 *`	

To cut a thread with more than 16 revolutions

`...`	
`N80 G01 Z-12.75 F1000 M3 *`	
`N90 G11 G41 H+180 R+32 F250 *`	
`N100 G26 R2 *`	Tangential approach

`N110 G98 L1 *`	Identify beginning of program section repeat
`N120 G13 G91 H+360 Z+1.5 F200 *`	Enter pitch directly as incremental Z value
`N130 L1.24 *`	Program the number of repeats (thread revolutions)
`N999999 %HELIX G71 *`	

7

Programming: Miscellaneous Functions

7.1 Entering Miscellaneous Functions M

Fundamentals

With the TNC's miscellaneous functions—also called M functions—you can influence:

- Program run, e.g., a program interruption
- Machine functions, such as switching spindle rotation and coolant supply on and off
- Contouring behavior of the tool

The machine tool builder may add some M functions that are not described in this User's Manual. Refer to your machine manual.

You can enter up to two M functions at the end of a positioning block.

You usually enter only the number of the M function. Some M functions can be programmed with additional parameters. In this case, the dialog is continued for the parameter input.

In the Manual Operation and Electronic Handwheel modes of operation, the M functions are entered with the M soft key.

Please note that some M functions become effective at the start of a positioning block, and others at the end.

M functions come into effect in the block in which they are called. Unless the M function is only effective blockwise, it is canceled in a subsequent block or at the end of the program. Some M functions are effective only in the block in which they are called.

7.2 Miscellaneous Functions for Program Run Control, Spindle and Coolant

Overview

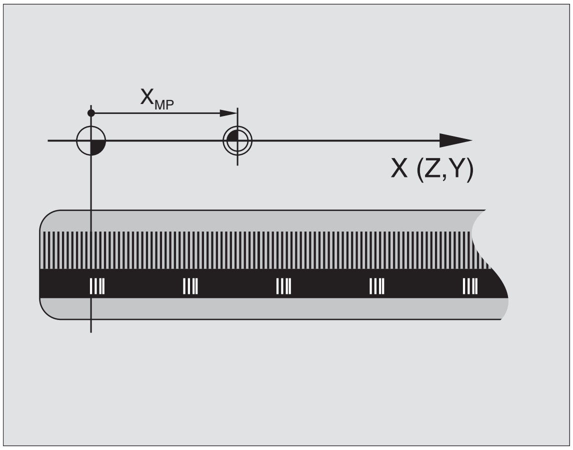
M	Effect	Effective at block	block start	end
M00	Stop program run Spindle STOP Coolant OFF			■
M01	Optional program STOP			■
M02	Stop program run Spindle STOP Coolant OFF Go to block 1 Clear the status display (dependent on Machine Parameter 7300)			■
M03	Spindle ON clockwise		■	
M04	Spindle ON counterclockwise		■	
M05	Spindle STOP			■
M06	Tool change Spindle STOP Program run stop (dependent on Machine Parameter 7440)			■
M08	Coolant ON		■	
M09	Coolant OFF			■
M13	Spindle ON clockwise Coolant ON		■	
M14	Spindle ON counterclockwise Coolant ON		■	
M30	Same as M02			■

7.3 Miscellaneous Functions for Coordinate Data

Programming machine-referenced coordinates: M91/M92

Scale reference point

On the scale, a reference mark indicates the position of the scale reference point.



Machine datum

The machine datum is required for the following tasks:

- Defining the limits of traverse (software limit switches)
- Moving to machine-referenced positions (such as tool change positions)
- Setting the workpiece datum

The distance in each axis from the scale reference point to the machine datum is defined by the machine tool builder in a machine parameter.

Standard behavior

The TNC references coordinates to the workpiece datum (see "Datum Setting (Without a 3-D Touch Probe)," page 24).

Behavior with M91—Machine datum

If you want the coordinates in a positioning block to be referenced to the machine datum, end the block with M91.

The coordinate values on the TNC screen are shown with respect to the machine datum. Switch the display of coordinates in the status display to REF (see "Status Displays," page 10).

Behavior with M92—Additional machine datum

In addition to the machine datum, the machine tool builder can also define an additional machine-based position as a reference point.

For each axis, the machine tool builder defines the distance between the machine datum and this additional machine datum. Refer to the machine manual for more information.

If you want the coordinates in a positioning block to be based on the additional machine datum, end the block with M92.

Radius compensation remains the same in blocks that are programmed with M91 or M92. The tool length, however, is **not** compensated.

Effect

M91 and M92 are effective only in the blocks in which they are programmed.

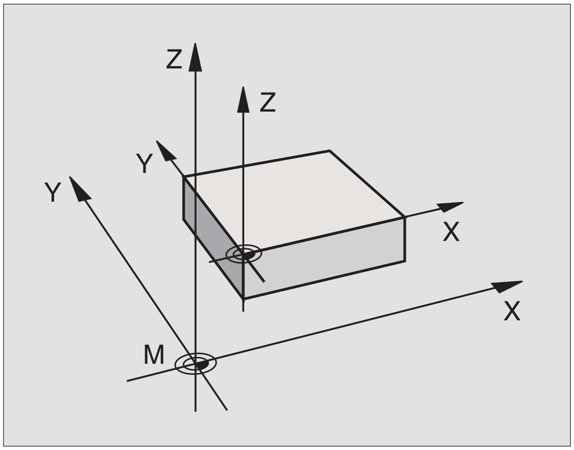
M91 and M92 take effect at the start of block.

Workpiece datum

If you want the coordinates to always be referenced to the machine datum, you can inhibit datum setting for one or more axes; (see "General User Parameters" on page 422).

If datum setting is inhibited for all axes, the TNC no longer displays the soft key DATUM SET in the Manual Operation mode.

The figure at right shows coordinate systems with the machine datum and workpiece datum.



M91/M92 in the Test Run mode

In order to be able to graphically simulate M91/M92 movements, you need to activate working space monitoring and display the workpiece blank referenced to the set datum, see "Showing the Workpiece in the Working Space (not TNC 410)," page 408.

Activating the most recently set datum: M104 (not with TNC 410)

Function

When processing pallet tables, the TNC may overwrite your most recently entered datum with values from the pallet table. With M104 you can reactivate the original datum.

Effect

M104 is effective only in the blocks in which it is programmed.

M104 becomes effective at the end of block.

Moving to positions in an untilted coordinate system with a tilted working plane: M130 (not with TNC 410)

Standard behavior with a tilted working plane

The TNC places the coordinates in the positioning blocks in the tilted coordinate system.

Behavior with M130

The TNC places coordinates in straight line blocks in the untilted coordinate system.

The TNC then positions the (tilted) tool to the programmed coordinates of the untilted system.

Following positioning blocks or fixed cycles are carried out in a tilted coordinate system. This can lead to problems in fixed cycles with absolute pre-positioning.
M130 is permitted only with a tilted plane.

Effect

M130 functions only in straight-line blocks without tool radius compensation and in blocks in which M130 is programmed.

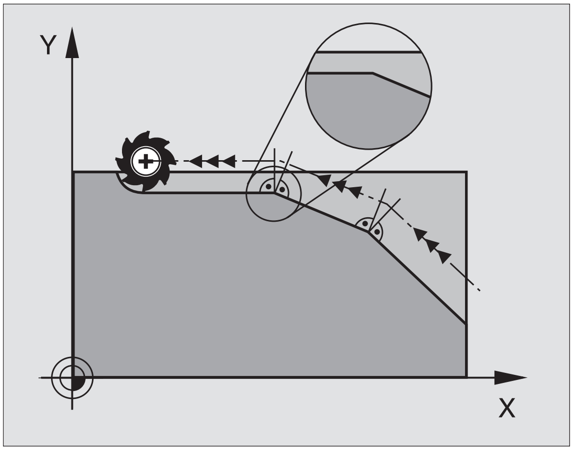
7.4 Miscellaneous Functions for Contouring Behavior

Smoothing corners: M90

Standard behavior

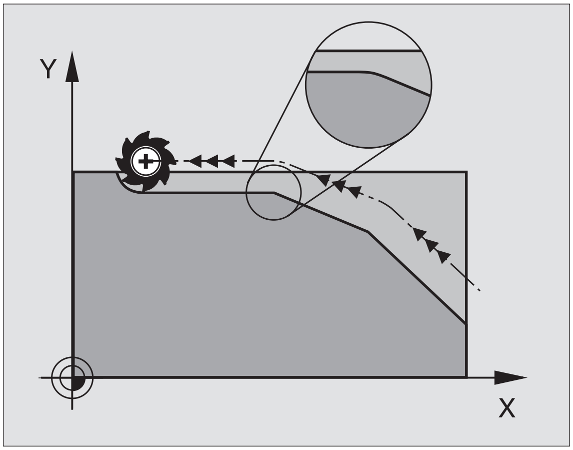
The TNC stops the tool briefly in positioning blocks without tool radius compensation. This is called an accurate stop.

In program blocks with radius compensation (**G41/G42**), the TNC automatically inserts a transition arc at outside corners.



Behavior with M90

The tool moves at corners with constant speed: This provides a smoother, more continuous surface. Machining time is also reduced. See figure at center right.



Application example: Surface consisting of a series of straight line segments.

Effect

M90 is effective only in the blocks in which it is programmed with M90.

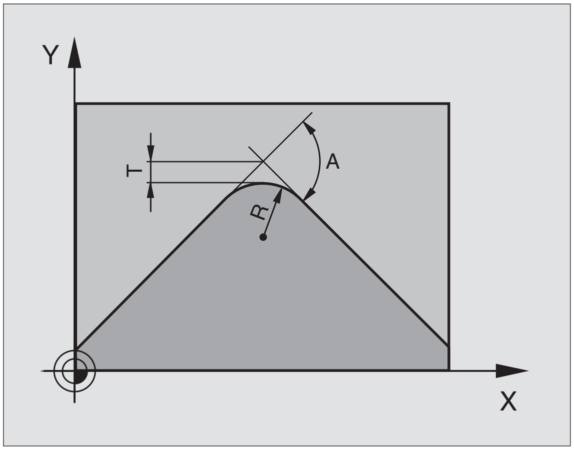
M90 becomes effective at the start of block. Operation with servo lag must be active.

> Independently of M90, you can use machine parameter MP7460 to set a limit value up to which the tool moves at constant path speed (effective with servo lag and feedforward control). Not with TNC 426 or TNC 430.

Insert rounding arc between straight lines: M112 (TNC 426, TNC 430)

Compatibility

For reasons of compatibility, the M112 function is still available on the TNC 426 and T30 controls. However, to define the tolerance for fast contour milling, HEIDENHAIN recommends the use of the TOLERANCE cycle for these TNCs, see “TOLERANCE (Cycle G62, not TNC 410),” page 313.



Entering contour transitions between contour elements: M112 (TNC 410)

Standard behavior

The TNC stops briefly for all changes in direction that are greater than the limit angle defined in MP7460(exact stop).

In program blocks with radius compensation (G41/G42), the TNC automatically inserts a transition arc at outside corners.

Behavior with M112

You can adjust the effect of M112 by redefining machine parameters.

The TNC inserts a selectable contour transition between any contour elements (compensated and uncompensated), in the plane or in three dimensions:

- Tangential circle: MP7415.0 = 0
  An acceleration jump results from the change in the curvature at the connection points.
- Third-degree polynomial (cubic spline): MP7415.0 = 1
  There is no velocity jump at the connection points.
- Fifth-degree polynomial: MP7415.0 = 2
  There is no acceleration jump at the connection points.
- Seventh-degree polynomial: MP7415.0 = 3 (standard setting)
  There is no jump in the rate of acceleration change

Permissible contour deviation E

With the tolerance value T you define the distance by which the milled contour can deviate from the programmed contour. If you do not enter a tolerance value, the TNC calculates the most exact contour transition possible at the programmed feed rate.

Limit angle H

If you enter a limit angle A, the TNC smoothens only those contour transitions whose angle of directional change is greater than the programmed limit angle. If you enter a limit angle = 0, the TNC moves the tool at a constant acceleration also over tangential transitions. Input range: 0° to 90°.

Entering M112 in a positioning block

If you press the soft key M112 in a positioning block (in answer to the "Miscellaneous function?" prompt), the TNC then continues the dialog by asking you for the permissible tolerance T and the limit angle A.

You can also define E and H through Q parameters, see “Principle and Overview,” page 330.

Effect

M112 is effective during operation with velocity feedforward as well as with servo lag.

M112 becomes effective at the start of block.

To cancel M112, enter M113.

Example NC block

```
N50 G01 G40 X+123.723 Y+25.491 F800 M112 E0.01 H10 *
```

Contour filter: M124 (not TNC 426, TNC 430)

Standard behavior

The TNC includes all available points in its calculation of a contour transition between contour elements.

Behavior with M124

You can adjust the effect of M124 by redefining machine parameters.

The TNC filters contour elements with small point spacing and inserts a transitional contour.

Shape of contour transition

- Tangential circle: MP7415.0 = 0
  An acceleration jump results from the change in the curvature at the connection points.
- Third-degree polynomial (cubic spline): MP7415.0 = 1
  There is no velocity jump at the connection points.
- Fifth-degree polynomial: MP7415.0 = 2
  There is no acceleration jump at the connection points.
- Seventh-degree polynomial: MP7415.0 = 3 (standard setting)
  There is no jump in the rate of acceleration change

Rounding of contour transitions

- Do not round the contour transition: MP7415.1 = 0
  Execute the contour transition as defined in MP7415.0 (standard contour transition: 7th-degree polynomial)
- Round the contour transition: MP7415.1 = 1
  Execute the contour transition so that the straight line segments remaining between the contour transitions are also rounded.

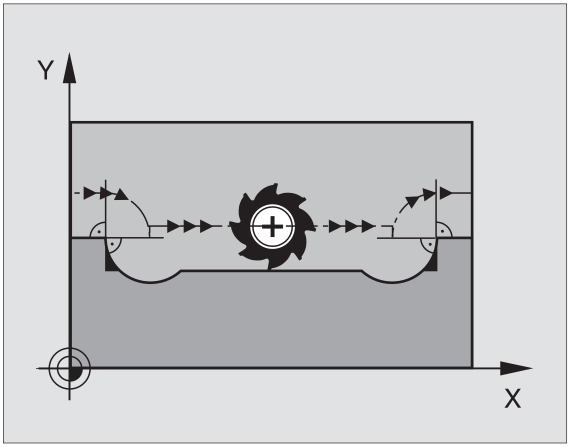
Minimum length E of a contour element

With parameter E you define the length up to which the TNC should filter contour elements out. If you have defined a permissible contour deviation in M112, the TNC will respect it. If you do not enter a maximum contour deviation, the TNC calculates the most exact contour transition possible without reducing the programmed feed rate.

Programming M124

If in a positioning block (with the dialog "Miscellaneous function") you press the soft key M124, the TNC then continues the dialog for this block and asks for the tolerance value E.

You can also define E through Q parameters, see “Principle and Overview,” page 330.



Effect

M124 becomes effective at the start of block. Like M112, M124 is reset with M113.

Example NC block

```
N50 G01 G40 X+123.723 Y+25.491 F800 M124 E0.01 *
```

Machining small contour steps: M97

Standard behavior

The TNC inserts a transition arc at outside corners. If the contour steps are very small, however, the tool would damage the contour.

In such cases the TNC interrupts program run and generates the error message “Tool radius too large.”

Behavior with M97

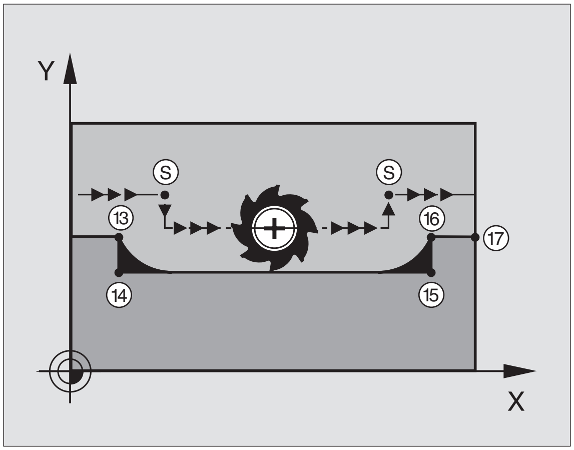
The TNC calculates the intersection of the contour elements—as at inside corners—and moves the tool over this point.

Program M97 in the same block as the outside corner.

Effect

M97 is effective only in the blocks in which it is programmed.

> A corner machined with M97 will not be completely finished. You may wish to rework the contour with a smaller tool.



Example NC blocks

`N50 G99 G01 ... R+20 *`	Large tool radius
`...`	
`N130 X ... Y ... F .. M97 *`	Move to contour point 13
`N140 G91 Y-0.5 .... F.. *`	Machine small contour step 13 to 14

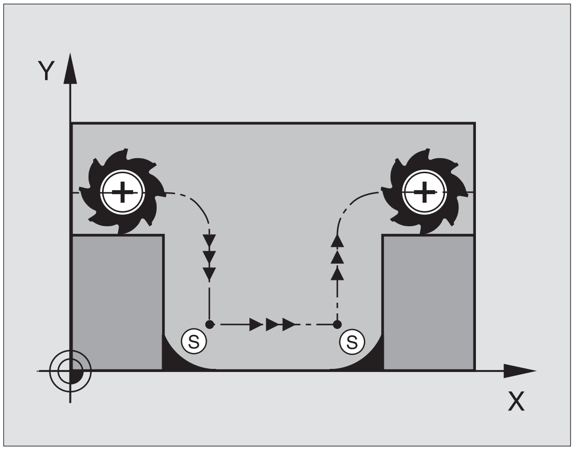
`N150 X+100 ... *`	Move to contour point 15
`N160 Y+0.5 ... F.. M97 *`	Machine small contour step 15 to 16
`N170 G90 X ... Y ... *`	Move to contour point 17

Machining open contours: M98

Standard behavior

The TNC calculates the intersections of the cutter paths at inside corners and moves the tool in the new direction at those points.

If the contour is open at the corners, however, this will result in incomplete machining.



Behavior with M98

With the miscellaneous function M98, the TNC temporarily suspends radius compensation to ensure that both corners are completely machined.

Effect

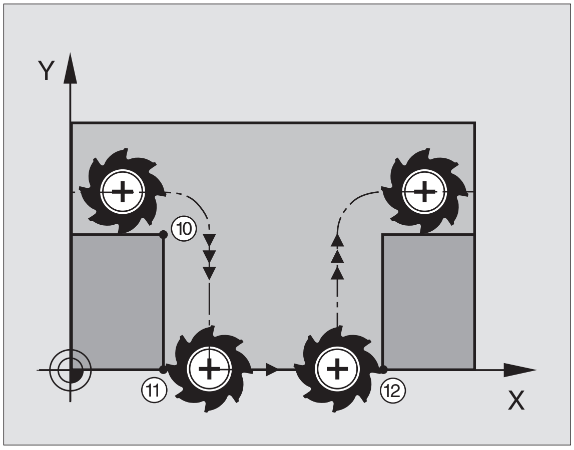
M98 is effective only in the blocks in which it is programmed.

M98 takes effect at the end of block.

Example NC blocks

Move to the contour points 10, 11 and 12 in succession:

```
N100 G01 G41 X ... Y... F... *
N110 X... G91 Y... M98 *
N120 X+ ... *
```



Feed rate factor for plunging movements: M103

Standard behavior

The TNC moves the tool at the last programmed feed rate, regardless of the direction of traverse.

Behavior with M103

The TNC reduces the feed rate when the tool moves in the negative direction of the tool axis. The feed rate for plunging FZMAX is calculated from the last programmed feed rate FPROG and a factor F%:

FZMAX = FPROG x F%

Programming M103

If you enter M103 in a positioning block, the TNC continues the dialog by asking you the factor F.

Effect

M103 becomes effective at the start of block.
To cancel M103, program M103 once again without a factor.

Example NC blocks

The feed rate for plunging is to be 20% of the feed rate in the plane.

`...`	Actual contouring feed rate (mm/min):
`N107 G01 G41 X+20 Y+20 F500 M103 F20 *`	500
`N180 Y+50 *`	500
`N190 G91 Z-2.5 *`	100
`N200 Y+5 Z-5 *`	141
`N210 X+50 *`	500
`N220 G90 Z+5 *`	500

Feed rate in millimeters per spindle revolution: M136 (not TNC 410)

Standard behavior

The TNC moves the tool at the programmed feed rate F in mm/min.

Behavior with M136

With M136, the TNC does not move the tool in mm/min, but rather at the programmed feed rate F in millimeters per spindle revolution. If you change the spindle speed by using the spindle override, the TNC changes the feed rate accordingly.

With the introduction of software 280 476-xx, the unit of measure used for miscellaneous function M136 has changed from µm/rev. to mm/rev. If you are using programs in which you have programmed M136 and which you have written on a previous TNC software, you need to reduce the value entered for the feed rate by the factor 1000.

Effect

M136 becomes effective at the start of block.

You can cancel M136 by programming M137.

Feed rate at circular arcs: M109/M110/M111

Standard behavior

The TNC applies the programmed feed rate to the path of the tool center.

Behavior at circular arcs with M109

The TNC adjusts the feed rate for circular arcs at inside and outside contours such that the feed rate at the tool cutting edge remains constant.

Behavior at circular arcs with M110

The TNC keeps the feed rate constant for circular arcs at inside contours only. At outside contours, the feed rate is not adjusted.

M110 is also effective for the inside machining of circular arcs using contour cycles.

Effect

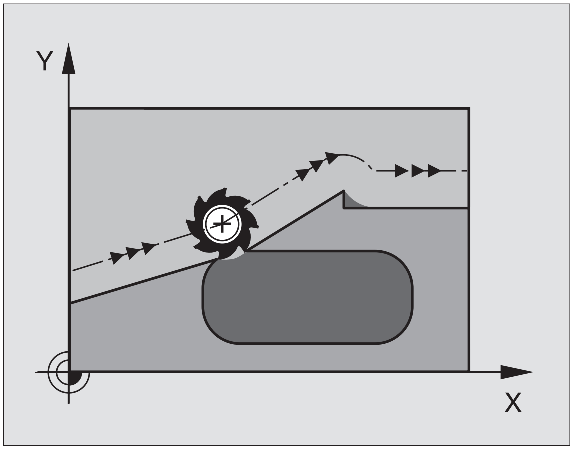
M109 and M110 become effective at the start of block.
To cancel M109 and M110, enter M111.

Calculating the radius-compensated path in advance (LOOK AHEAD): M120

Standard behavior

If the tool radius is larger than the contour step that is to be machined with radius compensation, the TNC interrupts program run and generates an error message. M97(see "Machining small contour steps: M97" on page 157): Although you can use M97 to inhibit the error message, this will result in dwell marks and will also move the corner.

If the programmed contour contains undercut features, the tool may damage the contour.



Behavior with M120

The TNC checks radius-compensated paths for contour undercuts and tool path intersections, and calculates the tool path in advance from the current block. Areas of the contour that might be damaged by the tool, are not machined (dark areas in figure at right). You can also use M120 to calculate the radius compensation for digitized data or data created on an external programming system. This means that deviations from the theoretical tool radius can be compensated.

Use LA (**L**ook **A**head) after M120 to define the number of blocks (maximum: 99) that you want the TNC to calculate in advance. Note that the larger the number of blocks you choose, the higher the block processing time will be.

Input

If you enter M120 in a positioning block, the TNC continues the dialog for this block by asking you the number of blocks LA that are to be calculated in advance.

Effect

M120 must be located in an NC block that also contains radius compensation G41 or G42. M120 is then effective from this block until

- radius compensation is canceled, or
- M120 LA0 is programmed, or
- M120 is programmed without LA.
- Call another program with %...

M120 becomes effective at the start of block.

Limitations

- After an external or internal stop, you can only re-enter the contour with the function RESTORE POS. AT N.
- If you are using the path functions G25 and G24, the blocks before and after G25 or CHF must contain only coordinates of the working plane.

Superimposing handwheel positioning during program run: M118 (not TNC 410)

Standard behavior

In the program run modes, the TNC moves the tool as defined in the part program.

Behavior with M118

M118 permits manual corrections by handwheel during program run. You can use this miscellaneous function by entering axis-specific values X, Y and Z (in mm) behind M118.

Programming M118

If you enter M118 in a positioning block, the TNC continues the dialog for this block by asking you the axis-specific values. The coordinates are entered with the orange axis direction buttons or the ASCII keyboard.

Effect

Cancel handwheel positioning by programming M118 once again without X, Y and Z.

M118 becomes effective at the start of block.

Example NC blocks

You wish to be able to use the handwheel during program run to move the tool in the working plane X/Y by ±1 mm of the programmed value:

```
G01 G41 X+0 Y+38.5 F125 M118 X1 Y1 *
```

M118 is always effective in the original coordinate system, even if the working plane is tilted.

M118 also functions in the Positioning with MDI mode of operation.

If M118 is active, the MANUAL OPERATION function is not available after a program interruption.

Erasing modal program information: M142 (not TNC 410)

Standard behavior

The TNC resets modal program information in the following situations:

- Select a new program.
- Execute a miscellaneous function M02, M30, or an N999999 %... block (depending on Machine Parameter 7300).
- Defining cycles for basic behavior with a new value

Behavior with M142

All modal program information except for basic rotation, 3-D rotation and Q parameters are reset.

Effect

M142 is effective only in the block in which it is programmed.

M142 becomes effective at the start of the block.

Erasing the basic rotation: M143 (not TNC 410)

Standard behavior

The basic rotation remains in effect until it is reset or is overwritten with a new value.

Behavior with M143

The TNC erases a programmed basic rotation from the NC program.

Effect

M143 is effective only in the block in which it is programmed.

M143 becomes effective at the start of the block.

7.5 Miscellaneous Functions for Rotary Axes

Feed rate in mm/min on rotary axes A, B, C: M116 (not TNC 410)

Standard behavior

The TNC interprets the programmed feed rate in a rotary axis in degrees per minute. The contouring feed rate therefore depends on the distance from the tool center to the center of the rotary axis.

The larger this distance becomes, the greater the contouring feed rate.

Feed rate in mm/min on rotary axes with M116

The machine geometry must be entered in Machine Parameters 7510 and following by the machine tool builder.

The TNC interprets the programmed feed rate in a rotary axis in mm/min. With this miscellaneous function, the TNC calculates the feed rate for each block at the start of the individual block. With a rotary axis, the feed rate is not changed during execution of the block even if the tool moves toward the center of the rotary axis.

Effect

M116 is effective in the working plane.
With M117 you can reset M116. M116 is also canceled at the end of the program.

M116 becomes effective at the start of block.

Shorter-path traverse of rotary axes: M126

Standard behavior

The standard behavior of the TNC while positioning rotary axes whose display has been reduced to values less than 360° is dependent on Machine Parameter 7682. In Machine Parameter 7682 is set whether the TNC should consider the difference between nominal and actual position, or whether the TNC should always (even without M126) choose the shortest path traverse to the programmed position. Examples:

Actual position	Nominal position	Traverse
350°	10°	–340°
10°	340°	+330°

Behavior with M126

With M126, the TNC will move the axis on the shorter path of traverse if you reduce display of a rotary axis to a value less than 360°. Examples:

Actual position	Nominal position	Traverse
350°	10°	+20°
10°	340°	–30°

Effect

M126 becomes effective at the start of block.
To cancel M126, enter M127. At the end of program, M126 is automatically canceled.

Reducing display of a rotary axis to a value less than 360°: M94

Standard behavior

The TNC moves the tool from the current angular value to the programmed angular value.

Example:

Current angular value:	538°
Programmed angular value:	180°
Actual distance of traverse:	–358°

Behavior with M94

At the start of block, the TNC first reduces the current angular value to a value less than 360° and then moves the tool to the programmed value. If several rotary axes are active, M94 will reduce the display of all rotary axes. As an alternative you can enter a rotary axis after M94. The TNC then reduces the display only of this axis.

Example NC blocks

To reduce display of all active rotary axes:

```
N50 M94 *
```

To reduce display of the C axis only

```
N50 M94 C *
```

To reduce display of all active rotary axes and then move the tool in the C axis to the programmed value:

```
N50 G00 C+180 M94 *
```

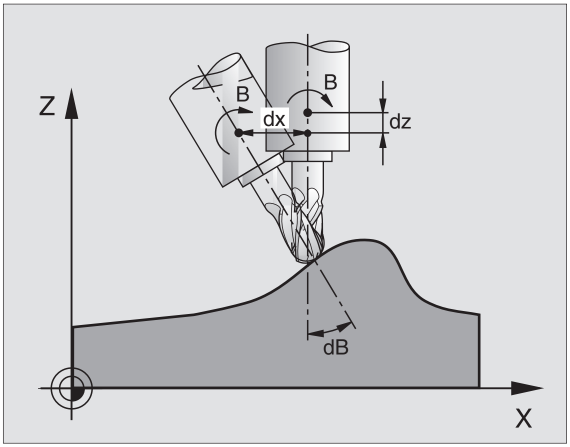
Effect

M94 is effective only in the block in which it is programmed.

M94 becomes effective at the start of block.

Automatic compensation of machine geometry when working with tilted axes: M114 (not TNC 410)

The machine geometry must be entered in Machine Parameters 7510 and following by the machine tool builder.



Standard behavior

The TNC moves the tool to the positions given in the part program. If the position of a tilted axis changes in the program, the resulting offset in the linear axes must be calculated by a postprocessor and traversed in a positioning block. As the machine geometry is also relevant, the NC program must be calculated separately for each machine tool.

Behavior with M114

If the position of a controlled tilted axis changes in the program, the TNC automatically compensates the tool offset by a 3-D length compensation. As the geometry of the individual machine tools is set in machine parameters, the TNC also compensates machine-specific offsets automatically. Programs only need to be calculated by the postprocessor once, even if they are being run on different machines with TNC control.

If your machine tool does not have controlled tilted axes (head tilted manually or positioned by the PLC), you can enter the current valid swivel head position after M114 (e.g. M114 B+45, Q parameters permitted).

The radius compensation must be calculated by a CAD system or by a postprocessor. A programmed radius compensation G41/G42 will result in an error message.

If the tool length compensation is calculated by the TNC, the programmed feed rate refers to the point of the tool. Otherwise it refers to the tool datum.

If your machine tool is equipped with a swivel head that can be tilted under program control, you can interrupt program run and change the position of the tilted axis, for example with the handwheel.

With the RESTORE POS. AT N function, you can then resume program run at the block at which the part program was interrupted. If M114 is active, the TNC automatically calculates the new position of the tilted axis.

If you wish to use the handwheel to change the position of the tilted axis during program run, use M118 in conjunction with M128.

Effect

M114 becomes effective at the start of block, M115 at the end of block. M114 is not effective when tool radius compensation is active.

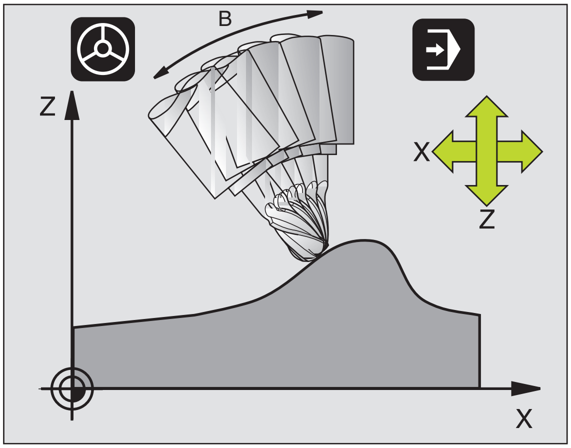
To cancel M114, enter M115. At the end of program, M114 is automatically canceled.

Maintaining the position of the tool tip when positioning with tilted axes (TCPM*): M128 (not TNC 410)

The machine geometry must be entered in Machine Parameters 7510 and following by the machine tool builder.

Standard behavior

The TNC moves the tool to the positions given in the part program. If the position of a tilted axis changes in the program, the resulting offset in the linear axes must be calculated and traversed in a positioning block (see figure with M114).



Behavior with M128

If the position of a controlled tilted axis changes in the program, the position of the tool tip to the workpiece remains the same.

If you wish to use the handwheel to change the position of the tilted axis during program run, use M118 in conjunction with M128. Handwheel positioning in a machine-based coordinate system is possible when M128 is active.

For tilted axes with Hirth coupling: Do not change the position of the tilted axis after retracting the tool. Otherwise you might damage the contour.

After M128 you can program another feed rate, at which the TNC will carry out the compensation movements in the linear axes. If you program no feed rate here, or if you program a larger feed rate than is defined in Machine Parameter 7471, the feed rate from Machine Parameter 7471 will be effective.

Reset M128 before positioning with M91 or M92 and before a T block.

To avoid contour gouging you must use only spherical cutters with M128.

The tool length must refer to the spherical center of the tool tip.

The TNC does not adjust the active radius compensation in accordance with the new position of the tilted axis. The result is an error which is dependent on the angular position of the rotary axis.

If M128 is active, the TNC shows in the status display the following symbol: .

M128 on tilting tables

If you program a tilting table movement while M128 is active, the TNC rotates the coordinate system accordingly. If for example you rotate the C axis by 90° (through a positioning command or datum shift) and then program a movement in the X axis, the TNC executes the movement in the machine axis Y.

The TNC also transforms the defined datum, which has been shifted by the movement of the rotary table.

M128 with 3-D tool compensation

If you carry out a 3-D tool compensation with active M128 and active radius compensation G41/G42, the TNC will automatically position the rotary axes for certain machine geometries (Peripheral milling, see "Peripheral Milling: 3-D Radius Compensation with Workpiece Orientation," page 115).

Effect

M128 becomes effective at the start of block, M129 at the end of block. M128 is also effective in the manual operating modes and remains active even after a change of mode. The feed rate for the compensation movement will be effective until you program a new feed rate or until you reset M128 with M129.

To cancel M128, enter M129. The TNC also resets M128 if you select a new program in a program run operating mode.

Example NC blocks

Moving at 1000 mm/min to compensate a radius.

```
G01 G41 X+0 Y+38.5 F125 M128 F1000 *
```

Exact stop at corners with nontangential transitions: M134 (not TNC 410)

Standard behavior

The standard behavior of the TNC during positioning with rotary axes is to insert a transitional element in nontangential contour transitions. The contour of the transitional element depends on the acceleration, the rate of acceleration (jerk), and the defined tolerance for contour deviation.

> With Machine Parameter 7440 you can change the standard behavior of the TNC so that M134 becomes active automatically whenever a program is selected, see "General User Parameters," page 422.

Behavior with M134

The TNC moves the tool during positioning with rotary axes so as to perform an exact stop at nontangential contour transitions.

Effect

M134 becomes effective at the start of block, M135 at the end of block.

You can reset M134 with M135. The TNC also resets M134 if you select a new program in a program run operating mode.

Selecting tilting axes: M138 (not TNC 410)

Standard behavior

The TNC performs M114 and M128, and tilts the working plane, only in those axes for which the machine tool builder has set the appropriate machine parameters.

Behavior with M138

The TNC performs the above functions only in those tilting axes that you have defined using M138.

Effect

M138 becomes effective at the start of block.

You can reset M138 by reprogramming it without entering any axes.

Example NC blocks

Perform the above-mentioned functions only in the tilting axis C:

```
G00 G40 Z+100 M138 C *
```

Compensating the machine's kinematic configuration for ACTUAL/NOMINAL positions at end of block: M144 (not TNC 410)

Standard behavior

The TNC moves the tool to the positions given in the part program. If the position of a tilted axis changes in the program, the resulting offset in the linear axes must be calculated, and traversed in a positioning block.

Behavior with M144

The TNC calculates into the position value any changes in the machine's kinematic configuration which result, for example, from adding a spindle attachment. If the position of a controlled tilted axis changes, the position of the tool tip to the workpiece is also changed. The resulting offset is calculated in the position display.

Positioning blocks with M91/M92 are permitted if M144 is active.

The position display in the operating modes FULL SEQUENCE and SINGLE BLOCK does not change until the tilting axes have reached their final position.

Effect

M144 becomes effective at the start of the block. M144 does not function in connection with M114, M128 or a tilted working plane.

You can cancel M144 by programming M145.

The machine geometry must be entered in Machine Parameters 7502 and following by the machine tool builder. The machine tool builder determines the behavior in the automatic and manual operating modes. Refer to your machine manual.

7.6 Miscellaneous Functions for Laser Cutting Machines (not TNC 410)

Principle

The TNC can control the cutting efficiency of a laser by transferring voltage values through the S-analog output. You can influence laser efficiency during program run through the miscellaneous functions M200 to M204.

Entering miscellaneous functions for laser cutting machines

If you enter an M function for laser cutting machines in a positioning block, the TNC continues the dialog by asking you the required parameters for the programmed function.

All miscellaneous functions for laser cutting machines become effective at the start of the block.

Output the programmed voltage directly: M200

Behavior with M200

The TNC outputs the value programmed after M200 as the voltage V.

Input range: 0 to 9 999 V

Effect

M200 remains in effect until a new voltage is output through M200, M201, M202, M203 or M204.

Output voltage as a function of distance: M201

Behavior with M201

M201 outputs the voltage in dependence on the distance to be covered. The TNC increases or decreases the current voltage linearly to the value programmed for V.

Input range: 0 to 9 999 V

Effect

M201 remains in effect until a new voltage is output through M200, M201, M202, M203 or M204.

Output voltage as a function of speed: M202

Behavior with M202

The TNC outputs the voltage as a function of speed. In the machine parameters, the machine tool builder defines up to three characteristic curves FNR in which specific feed rates are assigned to specific voltages. Use miscellaneous function M202 to select the curve FNR from which the TNC is to determine the output voltage.

Input range: 1 to 3

Effect

M202 remains in effect until a new voltage is output through M200, M201, M202, M203 or M204.

Output voltage as a function of time (time-dependent ramp): M203

Behavior with M203

The TNC outputs the voltage V as a function of the time TIME. The TNC increases or decreases the current voltage linearly to the value programmed for V within the time programmed for TIME.

Input range

Voltage V:	0 to 9.999 Volt
TIME:	0 to 1.999 seconds

Effect

M203 remains in effect until a new voltage is output through M200, M201, M202, M203 or M204.

Output voltage as a function of time (time-dependent pulse): M204

Behavior with M204

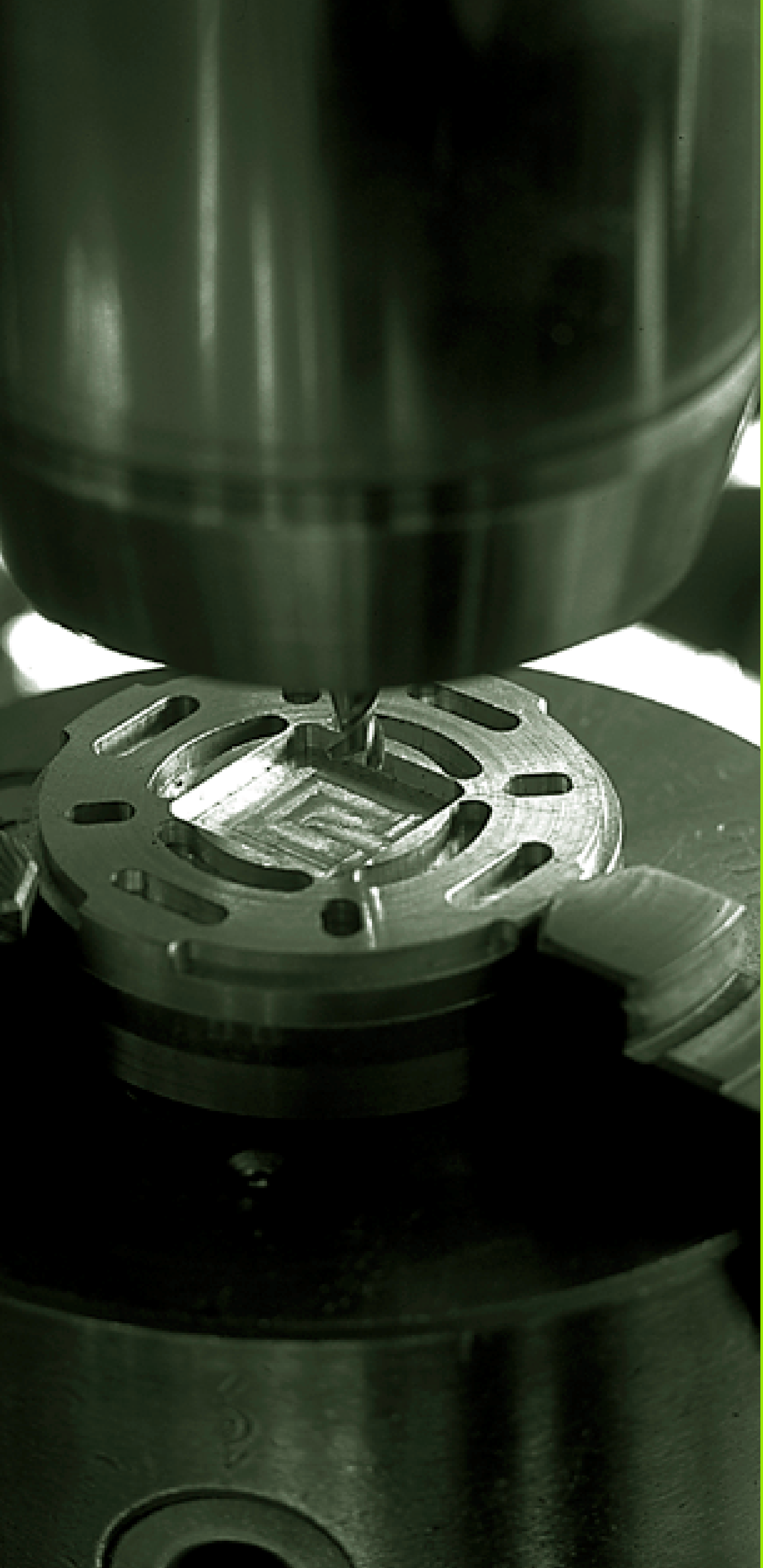
The TNC outputs a programmed voltage as a pulse with a programmed duration TIME.

Input range

Voltage V:	0 to 9.999 Volt
TIME:	0 to 1.999 seconds

Effect

M204 remains in effect until a new voltage is output through M200, M201, M202, M203 or M204.



8

Programming: Cycles

8.1 Working with Cycles

Frequently recurring machining cycles that comprise several working steps are stored in the TNC memory as standard cycles. Coordinate transformations and other special cycles are also provided as standard cycles (see table on next page).

Fixed cycles with numbers 200 and above use Q parameters as transfer parameters. Parameters with specific functions that are required in several cycles always have the same number: For example, Q200 is always assigned the set-up clearance, Q202 the plunging depth, etc.

Defining a cycle using soft keys

CYCL DEF

▶ The soft-key row shows the available groups of cycles.

DRILLING

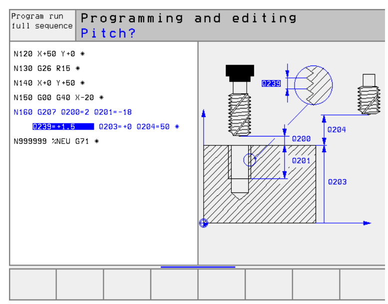
▶ Press the soft key for the desired group of cycles, for example DRILLING for the drilling cycles.

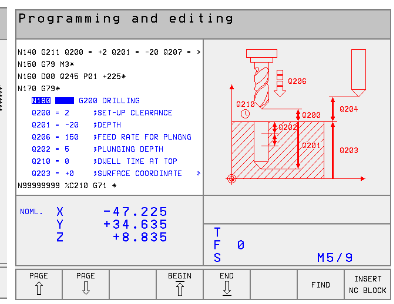
200

▶ Select a cycle, e.g. DRILLING. The TNC initiates the programming dialog and asks all required input values. At the same time a graphic of the input parameters is displayed in the right screen window. The parameter that is asked for in the dialog prompt is highlighted.

▶ Enter all parameters asked by the TNC and conclude each entry with the ENT key.

▶ The TNC terminates the dialog when all required data has been entered.





Example NC block

```
N10 G200 Q200=2 Q201=-20 Q206=150 Q202=5
    Q210=0 Q203=+0 Q204=50 Q211=0 *
```

Group of cycles	Soft key
Cycles for pecking, reaming, boring, counterboring, tapping and thread cutting	DRILLING
Cycles for milling pockets, studs and slots	POCKETS/ STUDS/ SLOTS
Cycles for producing hole patterns, such as circular or linear patterns	PATTERN
SL (Subcontour List) cycles which allow the contour-parallel machining of relatively complex contours consisting of several overlapping subcontours, cylinder surface interpolation (not TNC 410)	SL CYCLES
Cycles for face milling of flat or twisted surfaces	MULTIPASS MILLING
Coordinate transformation cycles which enable datum shift, rotation mirror image, enlarging and reducing for various contours	COORD. TRANSF.
Special cycles such as dwell time, program call, oriented spindle stop and tolerance (not TNC 410)	SPECIAL CYCLES

If you use indirect parameter assignments in fixed cycles with numbers greater than 200 (e.g. **`D00 Q210 = Q1`**), any change in the assigned parameter (e.g. Q1) will have no effect after the cycle definition. Define the cycle parameter (e.g. **`D00 Q210`**) directly in such cases.

In order to be able to run cycles G83 to G86, G74 to G78 and G56 to G59 on older TNC models, you must program an additional negative sign before the values for setup clearance and plunging depth.

Calling a cycle

Prerequisites

The following data must always be programmed before a cycle call:

- G30/G31 for graphic display (needed only for test graphics)
- Tool call
- Direction of spindle rotation (M functions M3/M4)
- Define cycle

For some cycles, additional prerequisites must be observed. They are described with the individual cycle.

The following cycles become effective automatically as soon as they are defined in the part program. These cycles cannot and must not be called:

- Cycle G220 for circular and Cycle G221 for linear hole patterns
- SL Cycle G14 CONTOUR GEOMETRY
- SL cycle G20 CONTOUR DATA (not TNC 410)
- Cycle G62 TOLERANCE (not TNC 410)
- Coordinate transformation cycles
- Cycle G04 DWELL TIME

All other cycles are called as described below:

1. If the TNC is to execute the cycle once after the last programmed block, program the cycle call with the miscellaneous functionM99 or with G79.
2. If the TNC is to execute the cycle automatically after every positioning block, program the cycle call with M89 (depending on machine parameter 7440).
3. only TNC 410: If the TNC is to execute the cycle at every position that is defined in a point table, use the function **G79 PAT** (see “Point Tables” on page 180).

To cancel M89, enter

- M99 or
- G79 or
- a new cycle.

Working with the secondary axes U/V/W

The TNC performs infeed movements in the axis that was defined in the TOOL CALL block as the spindle axis. It performs movements in the working plane only in the principal axes X, Y or Z. Exceptions:

- You program secondary axes for the side lengths in cycles G74 SLOT MILLING and G75/G76 POCKET MILLING.
- You program secondary axes in the contour geometry subprogram of an SL cycle.

8.2 Point Tables

Function

You should create a point table whenever you want to run a cycle, or several cycles in sequence, on an irregular point pattern.

If you are using drilling cycles, the coordinates of the working plane in the point table represent the hole centers. If you are using milling cycles, the coordinates of the working plane in the point table represent the starting-point coordinates of the respective cycle (e.g. center-point coordinates of a circular pocket). Coordinates in the spindle axis correspond to the coordinate of the workpiece surface.

Creating a point table

Select the **`Programming and Editing`** mode of operation.

PGM MGT

To call the file manager, press the PGM MGT key.

`FILE NAME ?`

NEW.PNT

ENT

Enter the name and file type of the point table and confirm your entry with the ENT key.

MM

To select the unit of measure, press the MM or INCH soft key. The TNC changes to the program blocks window and displays an empty point table.

INSERT LINE

With the soft key INSERT LINE, insert new lines and enter the coordinates of the desired machining position.

Repeat the process until all desired coordinates have been entered.

With the soft keys X OFF/ON, Y OFF/ON, Z OFF/ON (second soft-key row), you can specify which coordinates you want to enter in the point table.

Selecting a point table in the program

In the Programming and Editing mode of operation, select the program for which you want to activate the point table:

PGM CALL — Press the PGM CALL key to call the function for selecting the point table.

POINT TABLE — Press the POINT TABLE soft key.

Enter the name of the point table and confirm your entry with the ENT key.

Example NC block

```
N72 %:PAT: "NAMES"*
```

Calling a cycle in connection with point tables

With **G79 PAT** the TNC runs the point table that you last defined (even if you have defined the point table in a program that was nested with **%**).

The TNC uses the coordinate in the spindle axis as the clearance height for the cycle call.

If you want the TNC to call the last defined fixed cycle at the points defined in a point table, then program the cycle call with **G79 PAT**:

CYCL CALL

- To program the cycle call, press the CYCL CALL key.
- Press the CYCL CALL PAT soft key to call a point table.
- Enter the feed rate at which the TNC is to move from point to point (if you make no entry the TNC will move at the last programmed feed rate).
- If required, enter miscellaneous function M, then confirm with the END key.

The TNC moves the tool back to the clearance height over each successive starting point (clearance height = the spindle axis coordinate for cycle call). To use this procedure also for the cycles number 200 and greater, you must define the 2nd set-up clearance (Q204) as 0.

If you want to move at reduced feed rate when pre-positioning in the spindle axis, use the miscellaneous function M103 (see “Feed rate factor for plunging movements: M103” on page 158).

Effect of the point tables with Cycles G83, G84 and G74 to G78

The TNC interprets the points of the working plane as coordinates of the hole centers. The coordinate of the spindle axis defines the upper surface of the workpiece, so the TNC can pre-position automatically (first in the working plane, then in the spindle axis).

Effect of the point tables with SL Cycles and Cycle G39

The TNC interprets the points as an additional datum shift.

Effect of the point tables with Cycles G200 to G204

The TNC interprets the points of the working plane as coordinates of the hole centers. If you want to use the coordinate defined in the point table for the spindle axis as the starting point coordinate, you must define the workpiece surface coordinate (Q203) as 0.

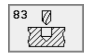
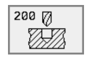
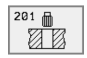
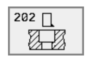
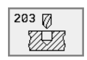
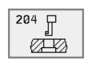
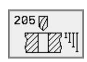
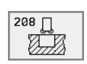
Effect of the point tables with Cycles 210 to 215

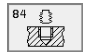
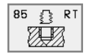
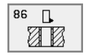
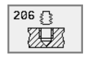
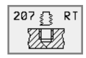
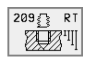
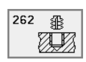
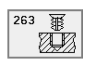
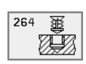
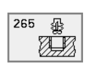
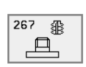
The TNC interprets the points as an additional datum shift. If you want to use the points defined in the point table as starting-point coordinates, you must define the starting points and the workpiece surface coordinate (Q203) in the respective milling cycle as 0.

8.3 Cycles for Drilling, Tapping and Thread Milling

Overview

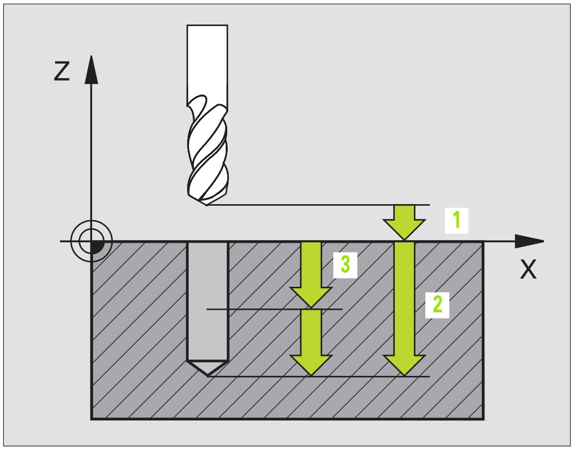
The TNC offers 9 (or 19) cycles for all types of drilling operations:

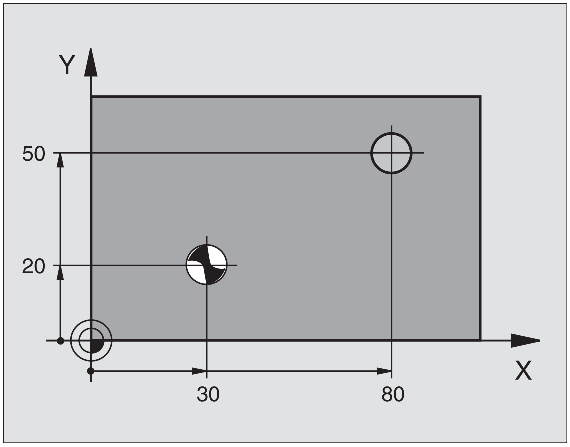
Cycle	Soft key
G83 PECKING Without automatic pre-positioning	
G200 DRILLING With automatic pre-positioning, 2nd set-up clearance	
G201 REAMING With automatic pre-positioning, 2nd set-up clearance	
G202 BORING With automatic pre-positioning, 2nd set-up clearance	
G203 UNIVERSAL DRILLING With automatic pre-positioning, 2nd set-up clearance, chip breaking, and decrementing	
G204 BACK BORING With automatic pre-positioning, 2nd set-up clearance	
G205 UNIVERSAL PECKING **(not TNC 410)** With automatic pre-positioning, 2nd set-up clearance, chip breaking, and advanced stop distance	
G208 BORE MILLING **(not TNC 410)** With automatic pre-positioning, 2nd set-up clearance	

Cycle	Soft key
G84 TAPPING With a floating tap holder	
G85 RIGID TAPPING Without a floating tap holder	
G86 THREAD CUTTING **(not TNC 410)**	
G206 TAPPING NEW **(not TNC 410)** With a floating tap holder, with automatic pre-positioning, 2nd set-up clearance	
G207 RIGID TAPPING NEW **(not TNC 410)** Without a floating tap holder, with automatic pre-positioning, 2nd set-up clearance	
G209 TAPPING W/ CHIP BRKG **(not TNC 410)** Without a floating tap holder, with automatic pre-positioning, 2nd set-up clearance, chip breaking	
G262 THREAD MILLING **(not TNC 410)** Cycle for milling a thread in pre-drilled material	
G263 THREAD MLLNG/CNTSNKG **(not TNC 410)** Cycle for milling a thread in pre-drilled material and machining a countersunk chamfer	
G264 THREAD DRILLING/MLLNG **(not TNC 410)** Cycle for drilling into the solid material with subsequent milling of the thread with a tool	
G265 HEL.THREAD DRLG/MLG **(not TNC 410)** Cycle for milling the thread into the solid material	
G267 OUTSIDE THREAD MLLNG **(not TNC 410)** Cycle for milling an external thread and machining a countersunk chamfer	

PECKING (Cycle G83)

1. The tool drills from the current position to the first plunging depth at the programmed feed rate F.
2. When it reaches the first plunging depth, the tool retracts at rapid traverse to the starting position and advances again to the first plunging depth minus the advanced stop distance t.
3. The advanced stop distance is automatically calculated by the control:
   - At a total hole depth of up to 30 mm: t = 0.6 mm
   - At a total hole depth exceeding 30 mm: t = hole depth / 50
   - Maximum advanced stop distance: 7 mm
4. The tool then advances with another infeed at the programmed feed rate F.
5. The TNC repeats this process (1 to 4) until the programmed depth is reached.
6. After a dwell time at the hole bottom, the tool is returned to the starting position at rapid traverse for chip breaking.



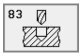


Before programming, note the following:

Program a positioning block for the starting point (hole center) in the working plane with radius compensation **G40.**

Program a positioning block for the starting point in the tool axis (set-up clearance above the workpiece surface).

The algebraic sign for the cycle parameter DEPTH determines the working direction. If you program DEPTH = 0, the cycle will not be executed.



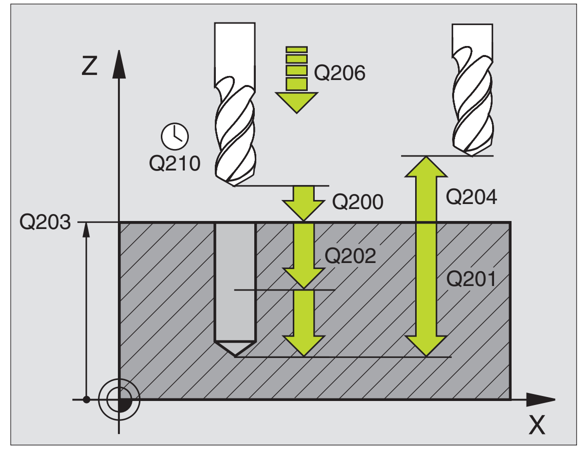
- **Set-up clearance 1** (incremental value): Distance between tool tip (at starting position) and workpiece surface
- **Total hole depth 2** (incremental value): Distance between workpiece surface and bottom of hole (tip of drill taper)
- **Plunging depth 3** (incremental value): Infeed per cut The total hole depth does not have to be a multiple of the plunging depth. The tool will drill to the total hole depth in one movement if:
  - the plunging depth is equal to the depth
  - the plunging depth is greater than the total hole depth
- **Dwell time in seconds**: Amount of time the tool remains at the total hole depth for chip breaking
- **Feed rate F**: Traversing speed of the tool during drilling in mm/min

Example: NC block

```
N10 G83 P01 2  P02 -20  P03 -8  P04 0
    P05 500*
```

DRILLING (Cycle G200)

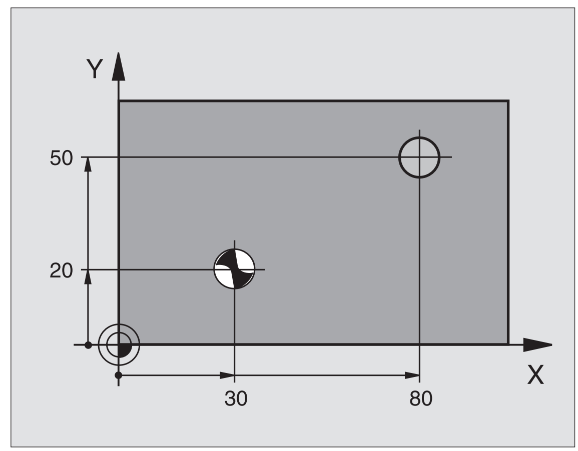
1 The TNC positions the tool in the tool axis at rapid traverse to the setup clearance above the workpiece surface.
2 The tool drills to the first plunging depth at the programmed feed rate F.
3 The TNC returns the tool at rapid traverse to the setup clearance, dwells there (if a dwell time was entered), and then moves at rapid traverse to the setup clearance above the first plunging depth.
4 The tool then advances with another infeed at the programmed feed rate F.
5 The TNC repeats this process (2 to 4) until the programmed depth is reached.
6 At the hole bottom, the tool is retracted to set-up clearance or, if programmed, to the 2nd set-up clearance at rapid traverse.

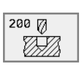


Before programming, note the following:

Program a positioning block for the starting point (hole center) in the working plane with radius compensation **G40.**

The algebraic sign for the cycle parameter DEPTH determines the working direction. If you program DEPTH = 0, the cycle will not be executed.





- **Set-up clearance** Q200 (incremental value): Distance between tool tip and workpiece surface. Enter a positive value.
- **Depth** Q201 (incremental value): Distance between workpiece surface and bottom of hole (tip of drill taper).
- **Feed rate for plunging** Q206: Traversing speed of the tool during drilling in mm/min.
- **Plunging depth** Q202 (incremental value): Infeed per cut. The depth does not have to be a multiple of the plunging depth. The TNC will go to depth in one movement if:
  - the plunging depth is equal to the depth
  - the plunging depth is greater than the depth

Example: NC block

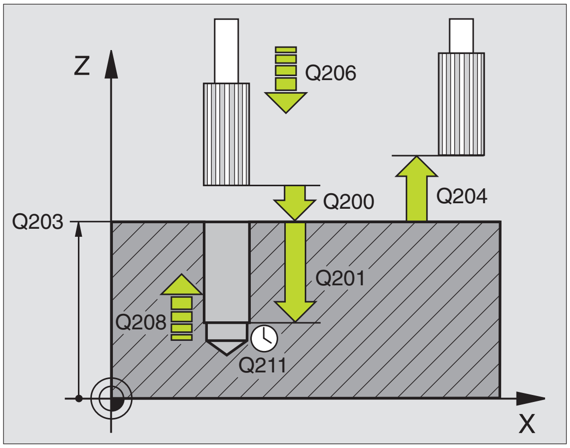
```
N70 G200 Q200=2 Q201=-20 Q206=150
    Q202=5 Q210=0 Q203=+0 Q204=50
    Q211=0 *
```

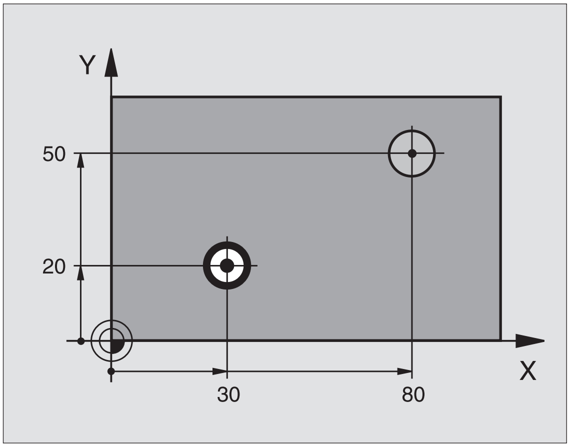
- **Dwell time at top** Q210: Time in seconds that the tool remains at set-up clearance after having been retracted from the hole for chip release.
- **Workpiece surface coordinate** Q203 (absolute value): Coordinate of the workpiece surface.
- **2nd set-up clearance** Q204 (incremental value): Coordinate in the tool axis at which no collision between tool and workpiece (clamping devices) can occur.

Not available with TNC 410:

- **Dwell time at depth** Q211: Time in seconds that the tool remains at the hole bottom.

REAMING (Cycle G201)



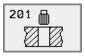


1. The TNC positions the tool in the tool axis at rapid traverse to the programmed setup clearance above the workpiece surface.
2. The tool reams to the entered depth at the programmed feed rate F.
3. If programmed, the tool remains at the hole bottom for the entered dwell time.
4. The tool then retracts to set-up clearance at the feed rate F, and from there—if programmed—to the 2nd set-up clearance at rapid traverse.

Before programming, note the following:

Program a positioning block for the starting point (hole center) in the working plane with radius compensation **G40.**

The algebraic sign for the cycle parameter DEPTH determines the working direction. If you program DEPTH = 0, the cycle will not be executed.



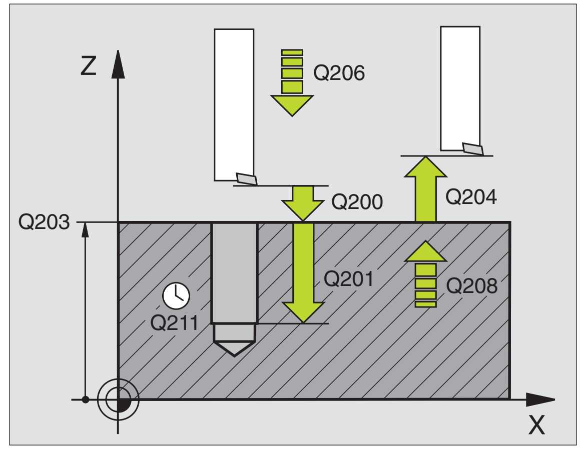
- **Set-up clearance** Q200 (incremental value): Distance between tool tip and workpiece surface.
- **Depth** Q201 (incremental value): Distance between workpiece surface and bottom of hole.
- **Feed rate for plunging** Q206: Traversing speed of the tool during reaming in mm/min.
- **Dwell time at depth** Q211: Time in seconds that the tool remains at the hole bottom.
- **Retraction feed rate** Q208: Traversing speed of the tool in mm/min when retracting from the hole. If you enter Q208 = 0, the tool retracts at the reaming feed rate.
- **Workpiece surface coordinate** Q203 (absolute value): Coordinate of the workpiece surface.
- **2nd set-up clearance** Q204 (incremental value): Coordinate in the tool axis at which no collision between tool and workpiece (clamping devices) can occur.

Example: NC block

```
N80 G201 Q200=2 Q201=-20 Q206=150
    Q211=0.25 Q208=30000 Q203=+0 Q204=50 *
```

BORING (Cycle G202)

The TNC and the machine tool must be specially prepared by the machine tool builder for the use of Cycle G202.



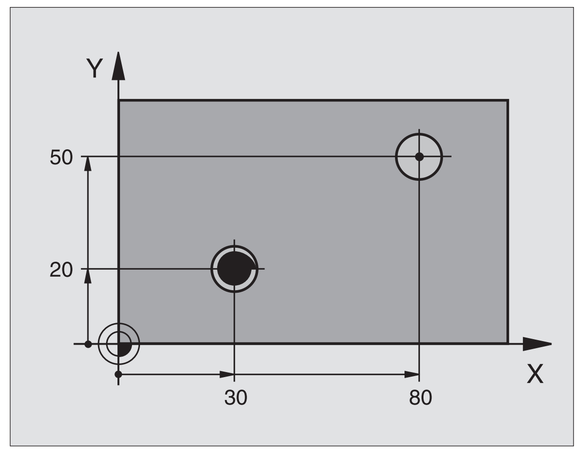
1 The TNC positions the tool in the tool axis at rapid traverse to the setup clearance above the workpiece surface.
2 The tool drills to the programmed depth at the feed rate for plunging.
3 If programmed, the tool remains at the hole bottom for the entered dwell time with active spindle rotation for cutting free.
4 The TNC then orients the spindle to the 0° position with an oriented spindle stop.
5 If retraction is selected, the tool retracts in the programmed direction by 0.2 mm (fixed value).
6 The TNC moves the tool at the retraction feed rate to the set-up clearance and then, if entered, to the 2nd set-up clearance at rapid traverse. If Q214=0, the tool point remains on the wall of the hole.

Before programming, note the following:

Program a positioning block for the starting point (hole center) in the working plane with radius compensation **G40.**

The algebraic sign for the cycle parameter DEPTH determines the working direction. If you program DEPTH = 0, the cycle will not be executed.

After the cycle is completed, the TNC restores the coolant and spindle conditions that were active before the cycle call.



- **Set-up clearance** Q200 (incremental value): Distance between tool tip and workpiece surface.
- **Depth** Q201 (incremental value): Distance between workpiece surface and bottom of hole.
- **Feed rate for plunging** Q206: Traversing speed of the tool during boring in mm/min.
- **Dwell time at depth** Q211: Time in seconds that the tool remains at the hole bottom.
- **Retraction feed rate** Q208: Traversing speed of the tool in mm/min when retracting from the hole. If you enter Q208 = 0, the tool retracts at feed rate for plunging.
- **Workpiece surface coordinate** Q203 (absolute value): Coordinate of the workpiece surface.

Example: NC block

```
N90 G202 Q200=2 Q201=-20 Q206=150
    Q211=0 Q208=30000 Q203=+0 Q204=50
    Q214=0 Q336=0 *
```

- **2nd set-up clearance** Q204 (incremental value): Coordinate in the tool axis at which no collision between tool and workpiece (clamping devices) can occur.
- **Disengaging direction** (0/1/2/3/4) Q214: Determine the direction in which the TNC retracts the tool at the hole bottom (after spindle orientation).

0: Do not retract tool
1: Retract tool in the negative reference axis direction
2: Retract tool in the negative secondary axis direction
3: Retract tool in the positive reference axis direction
4: Retract tool in the positive secondary axis direction

Danger of collision

Select a disengaging direction in which the tool moves away from the edge of the hole.

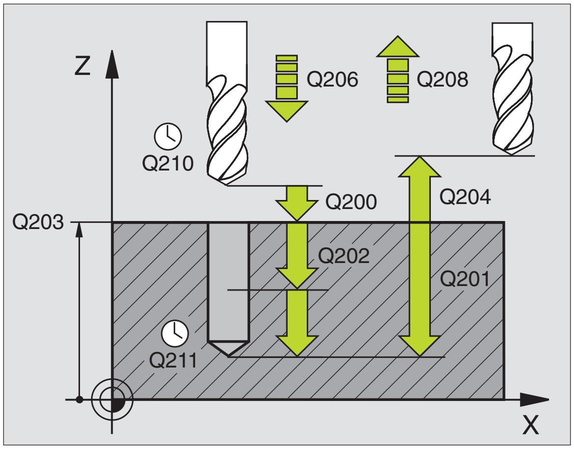
Check the position of the tool tip when you program a spindle orientation to the angle that you enter in Q336 (for example, in the Positioning with Manual Data Input mode of operation). Set the angle so that the tool tip is parallel to a coordinate axis.

Not available with TNC 410:

- **Angle for spindle orientation** Q336 (absolute value): Angle at which the TNC positions the tool before retracting it.

UNIVERSAL DRILLING (Cycle G203)

1 The TNC positions the tool in the tool axis at rapid traverse to the programmed setup clearance above the workpiece surface.
2 The tool drills to the first plunging depth at the programmed feed rate F.
3 If you have programmed chip breaking, the tool then retracts by the entered retraction value **(with TNC 410: by the set-up clearance)**. If you are working without chip breaking, the tool retracts at the retraction feed rate to set-up clearance, remains there—if programmed—for the entered dwell time, and advances again at rapid traverse to the set-up clearance above the first PLUNGING DEPTH.
4 The tool then advances with another infeed at the programmed feed rate. If programmed, the plunging depth is decreased after each infeed by the decrement.
5 The TNC repeats this process (2 to 4) until the programmed total hole depth is reached.
6 The tool remains at the hole bottom—if programmed—for the entered dwell time to cut free, and then retracts to set-up clearance at the retraction feed rate. If you have entered a 2nd set-up clearance, the tool subsequently moves to that position in rapid traverse.



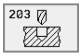
Example: NC block

```
N10 G203 Q200=2 Q201=-20 Q206=150
    Q202=5 Q210=0 Q203=+20 Q204=50
    Q212=0.2 Q213=3 Q205=3 Q211=0.25
    Q208=500 Q256=0.2 *
```

Before programming, note the following:

Program a positioning block for the starting point (hole center) in the working plane with radius compensation **G40.**

The algebraic sign for the cycle parameter DEPTH determines the working direction. If you program DEPTH = 0, the cycle will not be executed.



- **Set-up clearance** Q200 (incremental value): Distance between tool tip and workpiece surface.
- **Depth** Q201 (incremental value): Distance between workpiece surface and bottom of hole (tip of drill taper).
- **Feed rate for plunging** Q206: Traversing speed of the tool during drilling in mm/min.
- **Plunging depth** Q202 (incremental value): Infeed per cut. The depth does not have to be a multiple of the plunging depth. The TNC will go to depth in one movement if:
  - the plunging depth is equal to the depth
  - the plunging depth is greater than the depth
- **Dwell time at top** Q210: Time in seconds that the tool remains at set-up clearance after having been retracted from the hole for chip release.
- **Workpiece surface coordinate** Q203 (absolute value): Coordinate of the workpiece surface.

- **`2nd set-up clearance`** Q204 (incremental value): Coordinate in the tool axis at which no collision between tool and workpiece (clamping devices) can occur.
- **`Decrement`** Q212 (incremental value): Value by which the TNC decreases the plunging depth Q202 after each infeed.
- **`No. of breaks before retracting`** Q213: Number of chip breaks after which the TNC is to withdraw the tool from the hole for chip release. For chip breaking, the TNC retracts the tool each time by the value Q256 **(with TNC 410: by 0.2 mm)**.
- **`Minimum plunging depth`** Q205 (incremental value): If you have entered a decrement, the TNC limits the plunging depth to the value entered with Q205.
- **`Dwell time at depth`** Q211: Time in seconds that the tool remains at the hole bottom.
- **`Retraction feed rate`** Q208: Traversing speed of the tool in mm/min when retracting from the hole. If you enter Q208 = 0, the TNC retracts the tool at the feed rate in Q206.

Not available with TNC 410:

- **`Retraction rate for chip breaking`** Q256 (incremental value): Value by which the TNC retracts the tool during chip breaking.

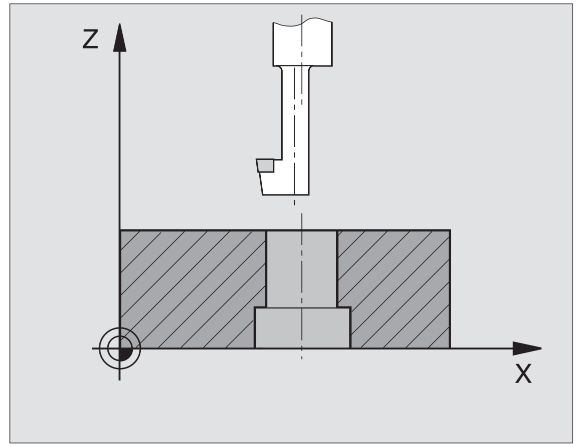
BACK BORING (Cycle G204)

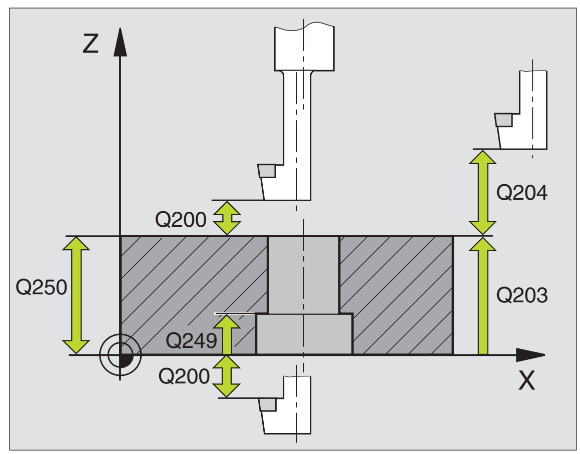
Machine and control must be specially prepared by the machine tool builder for use of this cycle.

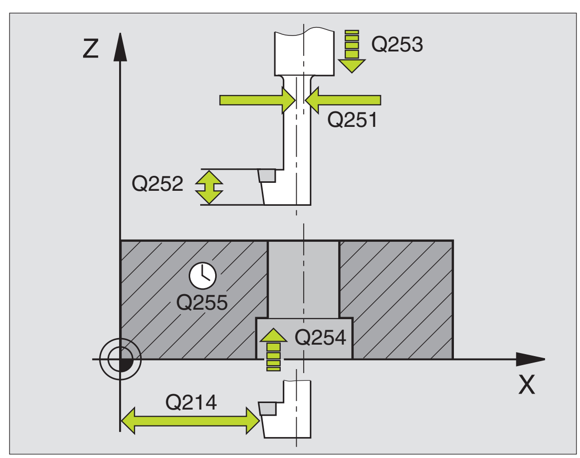
Special boring bars for upward cutting are required for this cycle.

This cycle allows holes to be bored from the underside of the workpiece.

1 The TNC positions the tool in the tool axis at rapid traverse to the setup clearance above the workpiece surface.
2 The TNC then orients the spindle to the 0° position with an oriented spindle stop, and displaces the tool by the off-center distance.
3 The tool is then plunged into the already bored hole at the feed rate for pre-positioning until the tooth has reached set-up clearance on the underside of the workpiece.
4 The TNC then centers the tool again over the bore hole, switches on the spindle and the coolant and moves at the feed rate for boring to the depth of bore.
5 If a dwell time is entered, the tool will pause at the top of the bore hole and will then be retracted from the hole again. The TNC carries out another oriented spindle stop and the tool is once again displaced by the off-center distance.
6 The TNC moves the tool at the pre-positioning feed rate to the setup clearance and then, if entered, to the 2nd setup clearance at rapid traverse.







Before programming, note the following:

Program a positioning block for the starting point (hole center) in the working plane with radius compensation **G40.**

The algebraic sign for the cycle parameter depth determines the working direction. Note: A positive sign bores in the direction of the positive spindle axis.

The entered tool length is the total length to the underside of the boring bar and not just to the tooth.

When calculating the starting point for boring, the TNC considers the tooth length of the boring bar and the thickness of the material.

- **`Set-up clearance`** Q200 (incremental value): Distance between tool tip and workpiece surface.
- **`Depth of counterbore`** Q249 (incremental value): Distance between underside of workpiece and the top of the hole. A positive sign means the hole will be bored in the positive spindle axis direction.
- **`Material thickness`** Q250 (incremental value): Thickness of the workpiece.
- **`Off-center distance`** Q251 (incremental value): Off-center distance for the boring bar; value from tool data sheet.
- **`Tool edge height`** Q252 (incremental value): Distance between the underside of the boring bar and the main cutting tooth; value from tool data sheet
- **`Feed rate for pre-positioning`** Q253: Traversing speed of the tool when moving in and out of the workpiece, in mm/min.
- **`Feed rate for counterboring`** Q254: Traversing speed of the tool during counterboring in mm/min.
- **`Dwell time`** Q255: Dwell time in seconds at the top of the bore hole.
- **`Workpiece surface coordinate`** Q203 (absolute value): Coordinate of the workpiece surface.
- **`2nd set-up clearance`** Q204 (incremental value): Coordinate in the tool axis at which no collision between tool and workpiece (clamping devices) can occur.
- **`Disengaging direction (0/1/2/3/4)`** Q214: Determine the direction in which the TNC displaces the tool by the off-center distance (after spindle orientation).

1: Displace tool in the negative reference axis direction
2: Displace tool in the negative secondary axis direction
3: Displace tool in the positive reference axis direction
4: Displace tool in the positive secondary axis direction

Danger of collision

Check the position of the tool tip when you program a spindle orientation to the angle that you enter in Q336 (for example, in the Positioning with Manual Data Input mode of operation). Set the angle so that the tool tip is parallel to a coordinate axis. Select a disengaging direction in which the tool moves away from the edge of the hole.

Example: NC block

```
N11 G204 Q200=2 Q249=+5 Q250=20 Q251=3.5
    Q252=15 Q253=750 Q254=200 Q255=0
    Q203=+20 Q204=50 Q214=1 Q336=0 *
```

Not available with TNC 410:

- **Angle for spindle orientation** Q336 (absolute value): Angle at which the TNC positions the tool before it is plunged into or retracted from the bore hole.

UNIVERSAL PECKING (Cycle G205, not TNC 410)

1. The TNC positions the tool in the tool axis at rapid traverse to the programmed setup clearance above the workpiece surface.
2. The tool drills to the first plunging depth at the programmed feed rate F.
3. If you have programmed chip breaking, the tool then retracts by the entered retraction value. If you are working without chip breaking, the tool is moved at rapid traverse to set-up clearance and then at rapid traverse to the entered starting position above the first plunging depth.
4. The tool then advances with another infeed at the programmed feed rate. If programmed, the plunging depth is decreased after each infeed by the decrement.
5. The TNC repeats this process (2 to 4) until the programmed total hole depth is reached.
6. The tool remains at the hole bottom—if programmed—for the entered dwell time to cut free, and then retracts to set-up clearance at the retraction feed rate. If you have entered a 2nd set-up clearance, the tool subsequently moves to that position in rapid traverse.

Before programming, note the following:

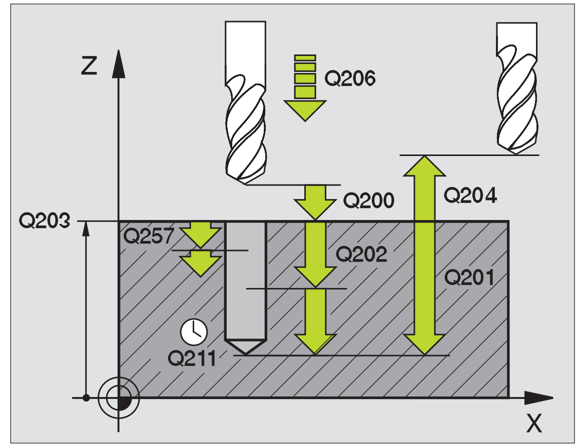
Program a positioning block for the starting point (hole center) in the working plane with radius compensation **G40.**

The algebraic sign for the cycle parameter DEPTH determines the working direction. If you program DEPTH = 0, the cycle will not be executed.

- **`Set-up clearance`** Q200 (incremental value): Distance between tool tip and workpiece surface.
- **`Depth`** Q201 (incremental value): Distance between workpiece surface and bottom of hole (tip of drill taper).
- **`Feed rate for plunging`** Q206: Traversing speed of the tool during drilling in mm/min.
- **`Plunging depth`** Q202 (incremental value): Infeed per cut. The depth does not have to be a multiple of the plunging depth. The TNC will go to depth in one movement if:
  - the plunging depth is equal to the depth
  - the plunging depth is greater than the depth
- **`Workpiece surface coordinate`** Q203 (absolute value): Coordinate of the workpiece surface.
- **`2nd set-up clearance`** Q204 (incremental value): Coordinate in the tool axis at which no collision between tool and workpiece (clamping devices) can occur.
- **`Decrement`** Q212 (incremental value): Value by which the TNC decreases the plunging depth Q202.
- **`Minimum plunging depth`** Q205 (incremental value): If you have entered a decrement, the TNC limits the plunging depth to the value entered with Q205.
- **`Upper advanced stop distance`** Q258 (incremental value): Set-up clearance for rapid traverse positioning when the TNC moves the tool again to the current plunging depth after retraction from the hole; value for the first plunging depth
- **`Lower advanced stop distance`** Q259 (incremental value): Set-up clearance for rapid traverse positioning when the TNC moves the tool again to the current plunging depth after retraction from the hole; value for the last plunging depth.

If you enter Q258 not equal to Q259, the TNC will change the advance stop distances between the first and last plunging depths at the same rate.

- **`Infeed depth for chip breaking`** Q257 (incremental value): Depth at which the TNC carries out chip breaking. There is no chip breaking if 0 is entered.
- **`Retraction rate for chip breaking`** Q256 (incremental value): Value by which the TNC retracts the tool during chip breaking.
- **`Dwell time at depth`** Q211: Time in seconds that the tool remains at the hole bottom.



Example: NC block

```
N12 G205 Q200=2 Q201=-80 Q206=150
    Q202=15 Q203=+100 Q204=50 Q212=0.5
    Q205=3 Q258=0.5 Q259=1 Q257=5
    Q256=0.2 Q211=0.25 *
```

BORE MILLING (Cycle G208, not TNC 410)

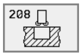
1. The TNC positions the tool in the tool axis at rapid traverse to the programmed set-up clearance above the workpiece surface and then moves the tool to the bore hole circumference on a rounded arc (if enough space is available).
2. The tool mills in a helix from the current position to the first plunging depth at the programmed feed rate.
3. When the drilling depth is reached, the TNC once again traverses a full circle to remove the material remaining after the initial plunge.
4. The TNC then positions the tool at the center of the hole again.
5. Finally the TNC returns to the setup clearance in rapid traverse. If you have entered a 2nd set-up clearance, the tool subsequently moves to that position in rapid traverse.

Before programming, note the following:

Program a positioning block for the starting point (hole center) in the working plane with radius compensation **G40.**

The algebraic sign for the cycle parameter DEPTH determines the working direction. If you program DEPTH = 0, the cycle will not be executed.

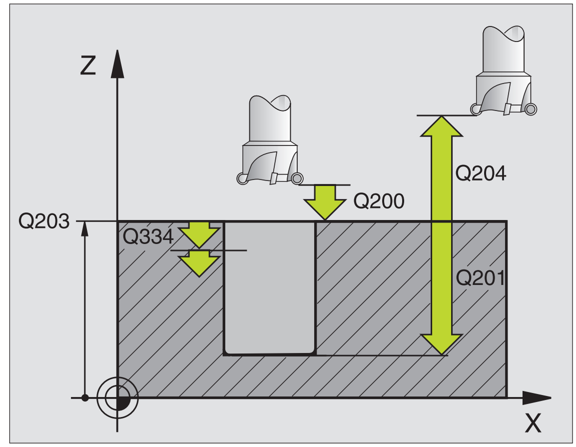
If you have entered the bore hole diameter to be the same as the tool diameter, the TNC will bore directly to the entered depth without any helical interpolation.

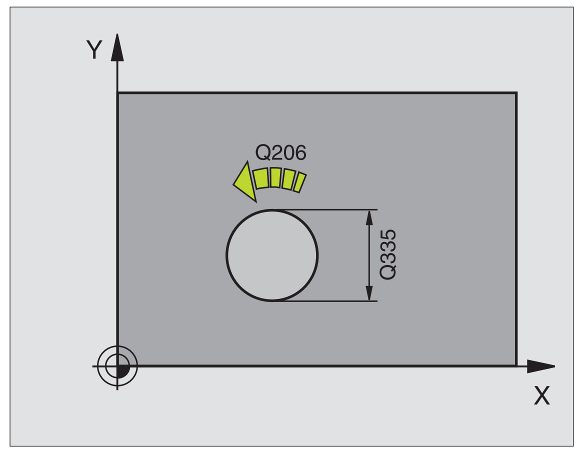


- **Set-up clearance** Q200 (incremental value): Distance between tool lower edge and workpiece surface.
- **Depth** Q201 (incremental value): Distance between workpiece surface and bottom of hole.
- **Feed rate for plunging** Q206: Traversing speed of the tool during helical drilling in mm/min.
- **Infeed per helix** Q334 (incremental value): Depth of the tool plunge with each helix (=360°).

> Note that if the infeed distance is too large, the tool or the workpiece may be damaged.
>
> To prevent the infeeds being too large, enter the max. plunge angle of the tool in the tool table, column **ANGLE**, see "Tool Data," page 99. The TNC then automatically calculates the max. infeed permitted and changes your entered value accordingly.

- **Workpiece surface coordinate** Q203 (absolute value): Coordinate of the workpiece surface.
- **2nd set-up clearance** Q204 (incremental value): Coordinate in the tool axis at which no collision between tool and workpiece (clamping devices) can occur.
- **Nominal diameter** Q335 (absolute value): Bore-hole diameter. If you have entered the nominal diameter to be the same as the tool diameter, the TNC will bore directly to the entered depth without any helical interpolation.
- **Roughing diameter** Q342 (absolute value): As soon as you enter a value greater than 0 in Q342, the TNC no longer checks the ratio between the nominal diameter and the tool diameter. This allows you to rough-mill holes whose diameter is more than twice as large as the tool diameter.



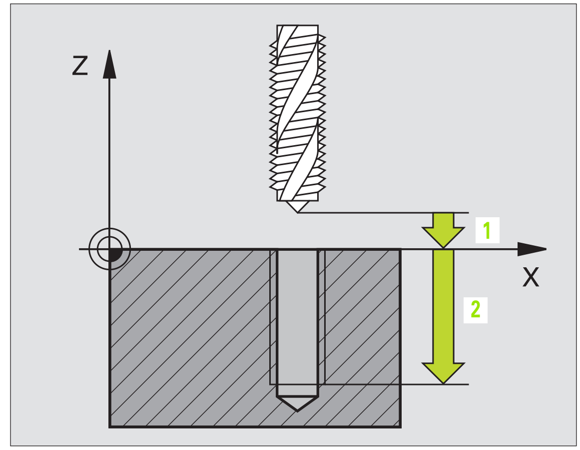


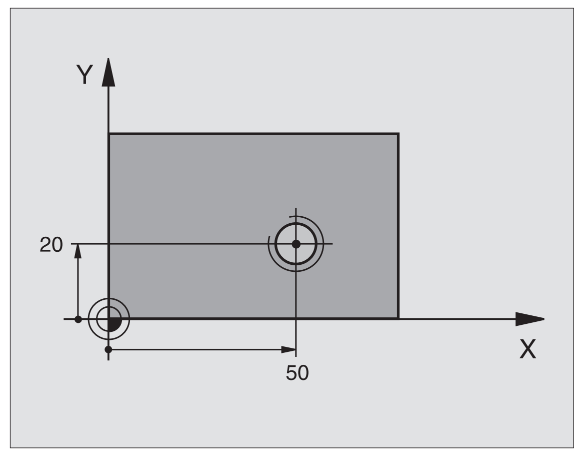
Example: NC block

```
N12 G208 Q200=2 Q201=-80 Q206=150
    Q334=1.5 Q203=+100 Q204=50 Q335=25
    Q342=0 *
```

TAPPING with a floating tap holder (Cycle G84)

1. The tool drills to the total hole depth in one movement.
2. Once the tool has reached the total hole depth, the direction of spindle rotation is reversed and the tool is retracted to the starting position at the end of the dwell time.
3. At the starting position, the direction of spindle rotation reverses once again.





Before programming, note the following:

Program a positioning block for the starting point (hole center) in the working plane with radius compensation **G40.**

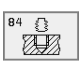
Program a positioning block for the starting point in the tool axis (set-up clearance above the workpiece surface).

The algebraic sign for the cycle parameter DEPTH determines the working direction. If you program DEPTH = 0, the cycle will not be executed.

A floating tap holder is required for tapping. It must compensate the tolerances between feed rate and spindle speed during the tapping process.

When a cycle is being run, the spindle speed override knob is disabled. The feed rate override knob is active only within a limited range, which is defined by the machine tool builder (refer to your machine manual).

For tapping right-hand threads activate the spindle with **M3**, for left-hand threads use **M4.**



- **Set-up clearance 1** (incremental value): Distance between tool tip (at starting position) and workpiece surface. Standard value: approx. 4 times the thread pitch
- **Total hole depth 2** (thread length, incremental value): Distance between workpiece surface and end of thread.
- **Dwell time in seconds**: Enter a value between 0 and 0.5 seconds to avoid wedging of the tool during retraction.
- **Feed rate F**: Traversing speed of the tool during tapping.

The feed rate is calculated as follows: F = S x p

F Feed rate (mm/min)
S: Spindle speed (rpm)
p: Thread pitch (mm)

Example: NC block

```
N13 G84 P01 2  P02 -20  P03 0  P04 100 *
```

Retracting after a program interruption

If you interrupt program run during tapping with the machine stop button, the TNC will display a soft key with which you can retract the tool.

TAPPING NEW with floating tap holder (Cycle G206, not TNC 410)

1. The TNC positions the tool in the tool axis at rapid traverse to the programmed setup clearance above the workpiece surface.
2. The tool drills to the total hole depth in one movement.
3. Once the tool has reached the total hole depth, the direction of spindle rotation is reversed and the tool is retracted to the set-up clearance at the end of the dwell time. If you have entered a 2nd set-up clearance, the tool subsequently moves to that position in rapid traverse.
4. At the set-up clearance, the direction of spindle rotation reverses once again.

Before programming, note the following:

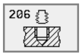
Program a positioning block for the starting point (hole center) in the working plane with radius compensation **G40.**

The algebraic sign for the cycle parameter DEPTH determines the working direction. If you program DEPTH = 0, the cycle will not be executed.

A floating tap holder is required for tapping. It must compensate the tolerances between feed rate and spindle speed during the tapping process.

When a cycle is being run, the spindle speed override knob is disabled. The feed rate override knob is active only within a limited range, which is defined by the machine tool builder (refer to your machine manual).

For tapping right-hand threads activate the spindle with **M3**, for left-hand threads use **M4.**



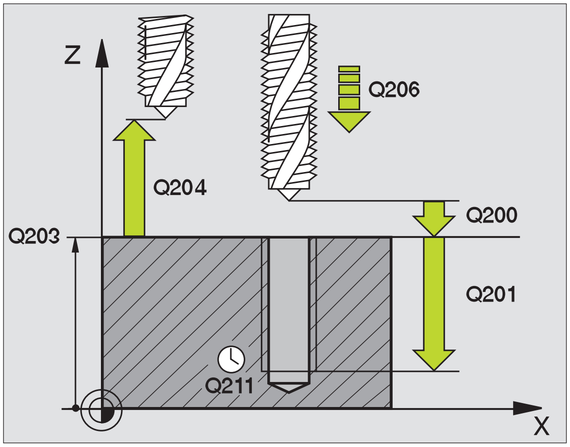
- **`Set-up clearance`** Q200 (incremental value): Distance between tool tip (at starting position) and workpiece surface. Standard value: approx. 4 times the thread pitch
- **`Total hole depth`** Q201 (thread length, incremental value): Distance between workpiece surface and end of thread.
- **`Feed rate F`** Q206: Traversing speed of the tool during tapping.
- **`Dwell time at bottom`** Q211: Enter a value between 0 and 0.5 seconds to avoid wedging of the tool during retraction.
- **`Workpiece surface coordinate`** Q203 (absolute value): Coordinate of the workpiece surface.
- **`2nd set-up clearance`** Q204 (incremental value): Coordinate in the tool axis at which no collision between tool and workpiece (clamping devices) can occur.

The feed rate is calculated as follows: F = S x p

F Feed rate (mm/min)
S: Spindle speed (rpm)
p: Thread pitch (mm)

Retracting after a program interruption

If you interrupt program run during tapping with the machine stop button, the TNC will display a soft key with which you can retract the tool.



Example: NC block

```
N25 G206 Q200=2 Q201=-20 Q206=150
    Q211=0.25 Q203=+25 Q204=50 *
```

RIGID TAPPING (Cycle G85)

Machine and control must be specially prepared by the machine tool builder for use of this cycle.

The TNC cuts the thread without a floating tap holder in one or more passes.

Rigid tapping offers the following advantages over tapping with a floating tap holder:

- Higher machining speeds possible.
- Repeated tapping of the same thread is possible; repetitions are enabled via spindle orientation to the 0° position during cycle call (depending on Machine Parameter 7160).
- Increased traverse range of the spindle axis due to absence of a floating tap holder.

Before programming, note the following:

Program a positioning block for the starting point (hole center) in the working plane with radius compensation **G40.**

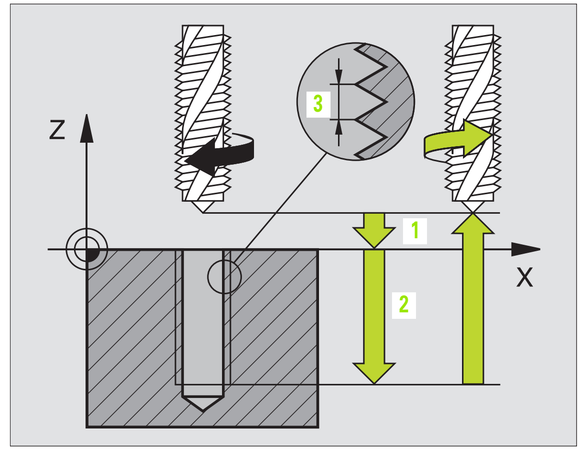
Program a positioning block for the starting point in the tool axis (set-up clearance above the workpiece surface).

The algebraic sign for the parameter total hole depth determines the working direction.

The TNC calculates the feed rate from the spindle speed. If the spindle speed override is used during tapping, the feed rate is automatically adjusted.

The feed-rate override knob is disabled.

At the end of the cycle the spindle comes to a stop. Before the next operation, restart the spindle with **M3** (or **M4**).



- **Set-up clearance 1** (incremental value): Distance between tool tip (at starting position) and workpiece surface
- **Total hole depth 2** (incremental value): Distance between workpiece surface (beginning of thread) and end of thread
- **Pitch 3**:
  Pitch of the thread. The algebraic sign differentiates between right-hand and left-hand threads:
  + = right-hand thread
  – = left-hand thread

Example: NC block

```
N18 G85 P01  2 P02 -20 P03 +1 *
```

Retract tool if program is interrupted (not TNC 410)

If you interrupt program run during tapping with the machine stop button, the TNC will display the soft key MANUAL OPERATION. If you press the MANUAL OPERATION key, you can retract the tool under program control. Simply press the positive axis direction button of the active tool axis.

RIGID TAPPING NEW (Cycle G207, not TNC 410)

Machine and control must be specially prepared by the machine tool builder for use of this cycle.

The TNC cuts the thread without a floating tap holder in one or more passes.

Rigid tapping offers the following advantages over tapping with a floating tap holder: See "RIGID TAPPING (Cycle G85)," page 202.

1. The TNC positions the tool in the tool axis at rapid traverse to the programmed setup clearance above the workpiece surface.
2. The tool drills to the total hole depth in one movement.
3. Once the tool has reached the total hole depth, the direction of spindle rotation is reversed and the tool is retracted to the set-up clearance at the end of the dwell time. If you have entered a 2nd set-up clearance, the tool subsequently moves to that position in rapid traverse.
4. The TNC stops the spindle turning at set-up clearance.

Before programming, note the following:

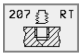
Program a positioning block for the starting point (hole center) in the working plane with radius compensation **G40**.

The algebraic sign for the parameter total hole depth determines the working direction.

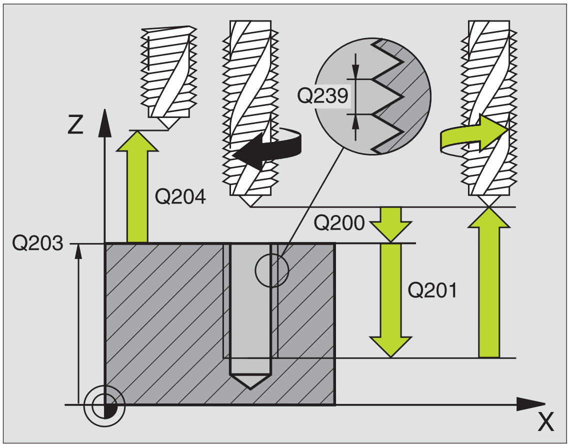
The TNC calculates the feed rate from the spindle speed. If the spindle speed override is used during tapping, the feed rate is automatically adjusted.

The feed-rate override knob is disabled.

At the end of the cycle the spindle comes to a stop. Before the next operation, restart the spindle with **M3** (or **M4**).



- **`Set-up clearance`** Q200 (incremental value): Distance between tool tip (at starting position) and workpiece surface.
- **`Total hole depth`** Q201 (incremental value): Distance between workpiece surface and end of thread.
- **`Pitch`** Q239
  Pitch of the thread. The algebraic sign differentiates between right-hand and left-hand threads:
  + = right-hand thread
  – = left-hand thread
- **`Workpiece surface coordinate`** Q203 (absolute value): Coordinate of the workpiece surface.
- **`2nd set-up clearance`** Q204 (incremental value): Coordinate in the tool axis at which no collision between tool and workpiece (clamping devices) can occur.



Example: NC block

```
N26 G207 Q200=2 Q201=-20 Q239=+1
    Q203=+25 Q204=50 *
```

Retracting after a program interruption

If you interrupt program run during thread cutting with the machine stop button, the TNC will display the MANUAL OPERATION soft key. If you press the MANUAL OPERATION key, you can retract the tool under program control. Simply press the positive axis direction button of the active tool axis.

THREAD CUTTING (Cycle G86, not TNC 410)

Machine and control must be specially prepared by the machine tool builder for use of this cycle.

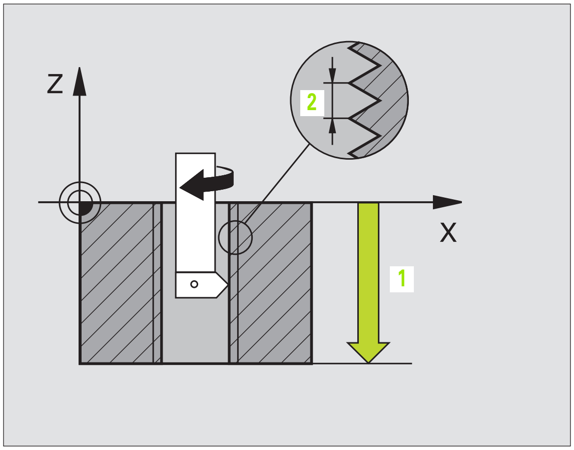
Cycle G86 THREAD CUTTING is performed by means of spindle control. The tool moves with the active spindle speed from its current position to the entered depth. As soon as it reaches the end of thread, spindle rotation is stopped. Tool approach and departure must be programmed separately. The most convenient way to do this is by using OEM cycles. The machine tool builder can give you further information.

Before programming, note the following:

The TNC calculates the feed rate from the spindle speed. If the spindle speed override is used during thread cutting, the feed rate is automatically adjusted.

The feed-rate override knob is disabled.

The TNC automatically activates and deactivates spindle rotation. Do not program **M3** or **M4** before cycle call.



Example: NC block

```
N22 G86 P01 -20 P02 +1 *
```

- **Total hole depth 1**: Distance between current tool position and end of thread

  The algebraic sign for the total hole depth determines the working direction (a negative value means a negative working direction in the tool axis)

- **Pitch 2**:
  Pitch of the thread. The algebraic sign differentiates between right-hand and left-hand threads:
  += right-hand thread (M3 with negative depth)
  – = left-hand thread (M4 with negative depth)

TAPPING WITH CHIP BREAKING (Cycle G209, not TNC 410)

Machine and control must be specially prepared by the machine tool builder for use of this cycle.

The tool machines the thread in several passes until it reaches the programmed depth. You can define in a parameter whether the tool is to be retracted completely from the hole for chip breaking.

1. The TNC positions the tool in the tool axis at rapid traverse to the programmed setup clearance above the workpiece surface. There it carries out an oriented spindle stop.
2. The tool moves to the programmed infeed depth, reverses the direction of spindle rotation and retracts by a specific distance or completely for chip release, depending on the definition.
3. It then reverses the direction of spindle rotation again and advances to the next infeed depth.
4. The TNC repeats this process (2 to 3) until the programmed thread depth is reached.
5. The tool is then retracted to set-up clearance. If you have entered a 2nd set-up clearance, the tool subsequently moves to that position in rapid traverse.
6. The TNC stops the spindle turning at set-up clearance.

Before programming, note the following:

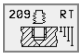
Program a positioning block for the starting point (hole center) in the working plane with radius compensation **G40**.

The algebraic sign for the parameter thread depth determines the working direction.

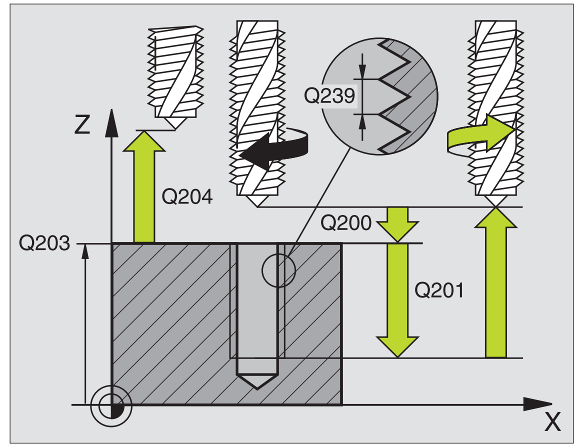
The TNC calculates the feed rate from the spindle speed. If the spindle speed override is used during tapping, the feed rate is automatically adjusted.

The feed-rate override knob is disabled.

At the end of the cycle the spindle comes to a stop. Before the next operation, restart the spindle with **M3** (or **M4**).



- **`Set-up clearance`** Q200 (incremental value): Distance between tool tip (at starting position) and workpiece surface.
- **`Thread depth`** Q201 (incremental value): Distance between workpiece surface and end of thread.
- **`Pitch`** Q239
  Pitch of the thread. The algebraic sign differentiates between right-hand and left-hand threads:
  + = right-hand thread
  – = left-hand thread
- **`Workpiece surface coordinate`** Q203 (absolute value): Coordinate of the workpiece surface.
- **`2nd set-up clearance`** Q204 (incremental value): Coordinate in the tool axis at which no collision between tool and workpiece (clamping devices) can occur.
- **`Infeed depth for chip breaking`** Q257 (incremental value): Depth at which TNC carries out chip breaking
- **`Retraction rate for chip breaking`** Q256: The TNC multiplies the pitch Q239 by the programmed value and retracts the tool by the calculated value during chip breaking. If you enter Q256 = 0, the TNC retracts the tool completely from the hole (to set-up clearance) for chip release.
- **`Angle for spindle orientation`** Q336 (absolute value): Angle at which the TNC positions the tool before machining the thread. This allows you to regroove the thread, if required.



Example: NC block

```
N26 G209 Q200=2 Q201=-20 Q239=+1
    Q203=+25 Q204=50 Q257=5 Q256=+25
    Q336=50 *
```

Retracting after a program interruption

If you interrupt program run during thread cutting with the machine stop button, the TNC will display the MANUAL OPERATION soft key. If you press the MANUAL OPERATION key, you can retract the tool under program control. Simply press the positive axis direction button of the active tool axis.

Fundamentals of thread milling

Prerequisites

- Your machine tool should feature internal spindle cooling (cooling lubricant min. 30 bar, compressed air supply min. 6 bar).
- Thread milling usually leads to distortions of the thread profile. To correct this effect, you need tool-specific compensation values which are given in the tool catalog or are available from the tool manufacturer. You program the compensation with the delta value for the tool radius DR in the tool call .
- The Cycles 262, 263, 264 and 267 can only be used with rightward rotating tools. For Cycle 265, you can use rightward and leftward rotating tools.
- The working direction is determined by the following input parameters: Algebraic sign Q239 (+ = right-hand thread /– = left-hand thread) and milling method Q351 (+1 = climb /–1 = up-cut). The table below illustrates the interrelation between the individual input parameters for rightward rotating tools.

Internal thread	Pitch	Climb/Up-cut	Work direction
Right-handed	+	+1(RL)	Z+
Left-handed	–	–1(RR)	Z+
Right-handed	+	–1(RR)	Z–
Left-handed	–	+1(RL)	Z–

External thread	Pitch	Climb/Up-cut	Work direction
Right-handed	+	+1(RL)	Z–
Left-handed	–	–1(RR)	Z–
Right-handed	+	–1(RR)	Z+
Left-handed	–	+1(RL)	Z+

Danger of collision

Always program the same algebraic sign for the infeeds: Cycles comprise several sequences of operation that are independent of each other. The order of precedence according to which the work direction is determined is described with the individual cycles. If you want to repeat specific machining operation of a cycle, for example with only the countersinking process, enter 0 for the thread depth. The work direction will then be determined from the countersinking depth.

Procedure in the case of a tool break

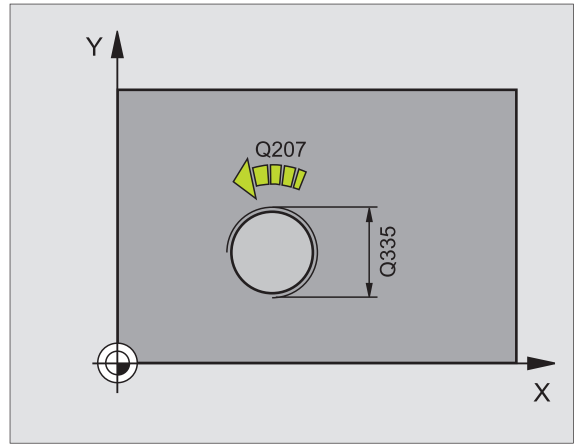
If a tool break occurs during thread cutting, stop the program run, change to the Positioning with MDI operating mode and move the tool in a linear path to the hole center. You can then retract the tool in the infeed axis and replace it.

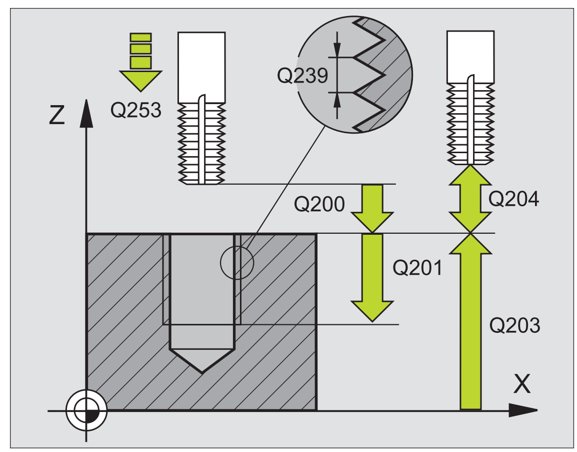
The TNC references the programmed feed rate during thread milling to the tool cutting edge. Since the TNC, however, always displays the feed rate relative to the path of the tool tip, the displayed value does not match the programmed value.

The machining direction of the thread changes if you execute a thread milling cycle in connection with Cycle 8 MIRRORING with only one axis.

THREAD MILLING (Cycle G262, not TNC 410)

1 The TNC positions the tool in the tool axis at rapid traverse to the programmed setup clearance above the workpiece surface.
2 The tool moves at the programmed feed rate for pre-positioning to the starting plane. The starting plane is derived from the algebraic sign of the thread pitch, the milling method (climb or up-cut milling) and the number of threads per step.
3 The tool then approaches the thread diameter tangentially in a helical movement. Before the helical approach, a compensating motion of the tool axis is carried out in order to begin at the programmed starting plane for the thread path.
4 Depending on the setting of the parameter for the number of threads, the tool mills the thread in one, in several spaced or in one continuous helical movement.
5 After this, the tool departs the contour tangentially and returns to the starting point in the working plane.
6 At the end of the cycle, the TNC retracts the tool in rapid traverse to set-up clearance, or—if programmed—to the 2nd set-up clearance.



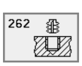


Before programming, note the following:

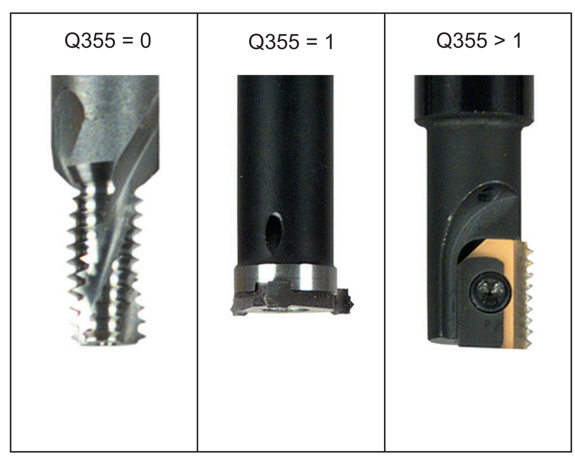
Program a positioning block for the starting point (hole center) in the working plane with radius compensation **G40.**

The algebraic sign for the cycle parameter thread depth determines the working direction. If you program the thread depth = 0, the cycle will not be executed.

The nominal thread diameter is approached in a semi-circle from the center. A pre-positioning movement to the side is carried out if the pitch of the tool diameter is four times smaller than the nominal thread diameter.



- **Nominal diameter** Q335: Nominal thread diameter.
- **Thread pitch** Q239: Pitch of the thread. The algebraic sign differentiates between right-hand and left-hand threads:
  + = right-hand thread
  – = left-hand thread
- **Thread depth** Q201 (incremental value): Distance between workpiece surface and root of thread.
- **Threads per step** Q355: Number of thread revolutions by which the tool is offset, see figure at lower right
  0 = one 360° helical path to the depth of thread.
  1 = continuous helical path over the entire length of the thread
  >1 = several helical paths with approach and departure; between them, the TNC offsets the tool by Q355, multiplied by the pitch



- **`Feed rate for pre-positioning`** Q253: Traversing speed of the tool when moving in and out of the workpiece, in mm/min.
- **`Climb or up-cut`** Q351: Type of milling operation with M03
  +1 = climb milling
  –1 = up-cut milling
- **`Set-up clearance`** Q200 (incremental value): Distance between tool tip and workpiece surface.
- **`Workpiece surface coordinate`** Q203 (absolute value): Coordinate of the workpiece surface.
- **`2nd set-up clearance`** Q204 (incremental value): Coordinate in the tool axis at which no collision between tool and workpiece (clamping devices) can occur.
- **`Feed rate for milling`** Q207: Traversing speed of the tool in mm/min while milling.

Example: NC block

```
N25 G262 Q335=10 Q239=+1.5 Q201=-20
    Q335=0 Q253=750 Q351=+1 Q200=2
    Q203=+30 Q204=50 Q207=500 *
```

THREAD MILLING/COUNTERSINKING (Cycle G263, not TNC 410)

1 The TNC positions the tool in the tool axis at rapid traverse to the programmed setup clearance above the workpiece surface.

Countersinking

2 The tool moves at the feed rate for pre-positioning to the countersinking depth minus the setup clearance, and then at the feed rate for countersinking to the countersinking depth.

3 If a safety clearance to the side has been entered, the TNC immediately positions the tool at the feed rate for pre-positioning to the countersinking depth

4 Then, depending on the available space, the TNC makes a tangential approach to the core diameter, either tangentially from the center or with a pre-positioning move to the side, and follows a circular path.

Countersinking at front

5 The tool moves at the feed rate for pre-positioning to the sinking depth at front.

6 The TNC positions the tool without compensation from the center on a semicircle to the offset at front, and then follows a circular path at the feed rate for countersinking.

7 The tool then moves in a semicircle to the hole center.

Thread milling

8 The TNC moves the tool at the programmed feed rate for pre-positioning to the starting plane for the thread. The starting plane is determined from the thread pitch and the type of milling (climb or up-cut).

9 Then the tool moves tangentially on a helical path to the thread diameter and mills the thread with a 360° helical motion.

10 After this, the tool departs the contour tangentially and returns to the starting point in the working plane.

11 At the end of the cycle, the TNC retracts the tool in rapid traverse to set-up clearance, or—if programmed—to the 2nd set-up clearance.

Before programming, note the following:

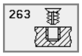
Program a positioning block for the starting point (hole center) in the working plane with radius compensation **G40.**

The algebraic sign of the cycle parameters depth of thread, countersinking depth or sinking depth at front determines the working direction. The working direction is defined in the following sequence:
1st: Depth of thread
2nd: Countersinking depth
3rd: Depth at front

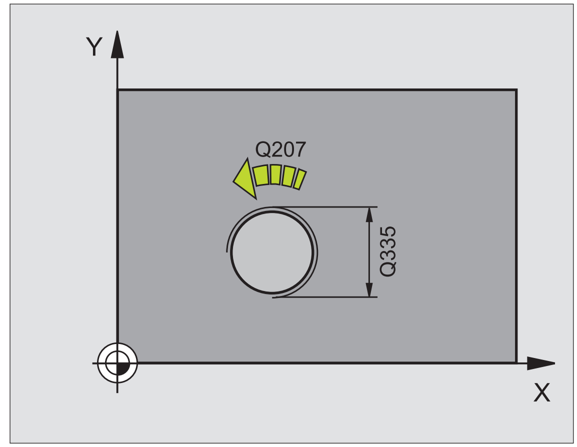
If you program a depth parameter to be 0, the TNC does not execute that step.

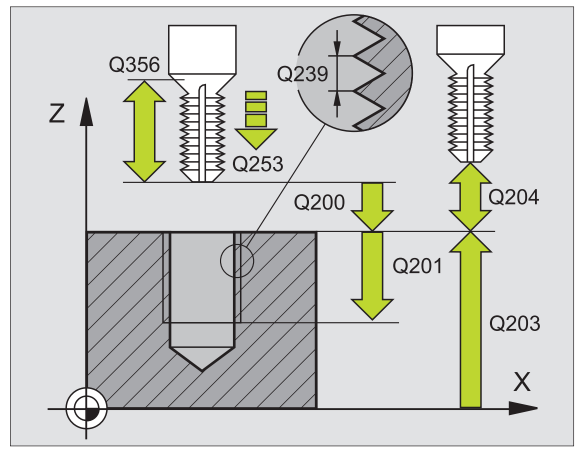
If you wish to countersink with the front of the tool, define the countersinking depth as 0.

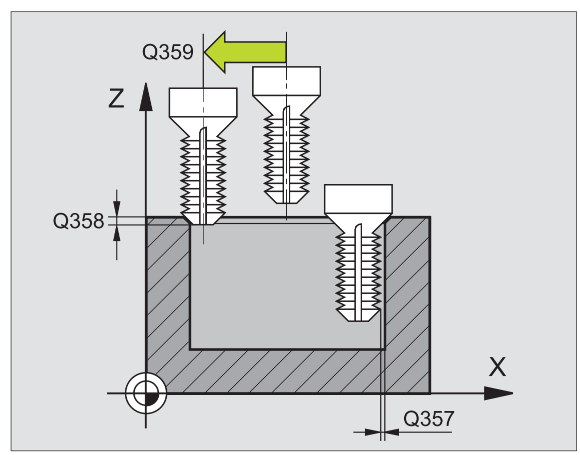
Program the thread depth as a value smaller than the countersinking depth by at least one-third the thread pitch.



- **`Nominal diameter`** Q335: Nominal thread diameter.
- **`Thread pitch`** Q239: Pitch of the thread. The algebraic sign differentiates between right-hand and left-hand threads:
  + = right-hand thread
  – = left-hand thread
- **`Thread depth`** Q201 (incremental value): Distance between workpiece surface and root of thread.
- **`Countersinking depth`** Q356 (incremental value): Distance between tool point and the top surface of the workpiece.
- **`Feed rate for pre-positioning`** Q253: Traversing speed of the tool when moving in and out of the workpiece, in mm/min.
- **`Climb or up-cut`** Q351: Type of milling operation with M03
  +1 = climb milling
  –1 = up-cut milling
- **`Set-up clearance`** Q200 (incremental value): Distance between tool tip and workpiece surface.
- **`Set-up clearance to the side`** Q357 (incremental value): Distance between tool tooth and the wall.
- **`Depth at front`** Q358 (incremental value): Distance between tool point and the top surface of the workpiece for countersinking at the front of the tool.
- **`Countersinking offset at front`** Q359 (incremental value): Distance by which the TNC moves the tool center away from the hole center.







- **Workpiece surface coordinate** Q203 (absolute value): Coordinate of the workpiece surface.
- **2nd set-up clearance** Q204 (incremental value): Coordinate in the tool axis at which no collision between tool and workpiece (clamping devices) can occur.
- **Feed rate for counterboring** Q254: Traversing speed of the tool during counterboring in mm/min.
- **Feed rate for milling** Q207: Traversing speed of the tool in mm/min while milling.

Example: NC block

```
N25 G263 Q335=10 Q239=+1.5 Q201=-16
    Q356=-20 Q253=750 Q351=+1 Q200=2
    Q357=0.2 Q358=+0 Q359=+0 Q203=+30
    Q204=50 Q254=150 Q207=500 *
```

THREAD DRILLING/MILLING (Cycle G264) not TNC 410)

1 The TNC positions the tool in the tool axis at rapid traverse to the programmed setup clearance above the workpiece surface.

Drilling

2 The tool drills to the first plunging depth at the programmed feed rate for plunging.
3 If you have programmed chip breaking, the tool then retracts by the entered retraction value. If you are working without chip breaking, the tool is moved at rapid traverse to set-up clearance and then at rapid traverse to the entered starting position above the first plunging depth.
4 The tool then advances with another infeed at the programmed feed rate.
5 The TNC repeats this process (2 to 4) until the programmed total hole depth is reached.

Countersinking at front

6 The tool moves at the feed rate for pre-positioning to the sinking depth at front.
7 The TNC positions the tool without compensation from the center on a semicircle to the offset at front, and then follows a circular path at the feed rate for countersinking.
8 The tool then moves in a semicircle to the hole center.

Thread milling

9 The TNC moves the tool at the programmed feed rate for pre-positioning to the starting plane for the thread. The starting plane is determined from the thread pitch and the type of milling (climb or up-cut).
10 Then the tool moves tangentially on a helical path to the thread diameter and mills the thread with a 360° helical motion.
11 After this, the tool departs the contour tangentially and returns to the starting point in the working plane.

12 At the end of the cycle, the TNC retracts the tool in rapid traverse to set-up clearance, or—if programmed—to the 2nd set-up clearance.

Before programming, note the following:

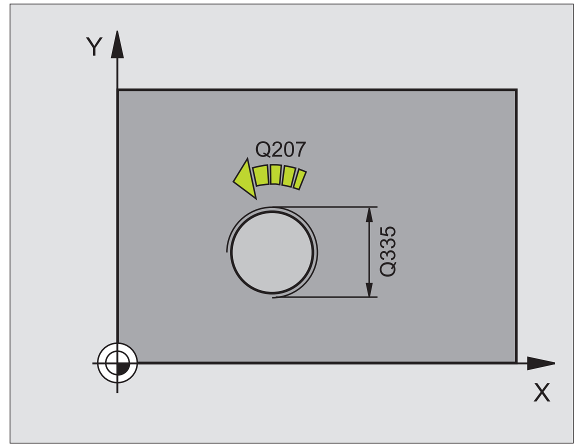
Program a positioning block for the starting point (hole center) in the working plane with radius compensation **G40.**

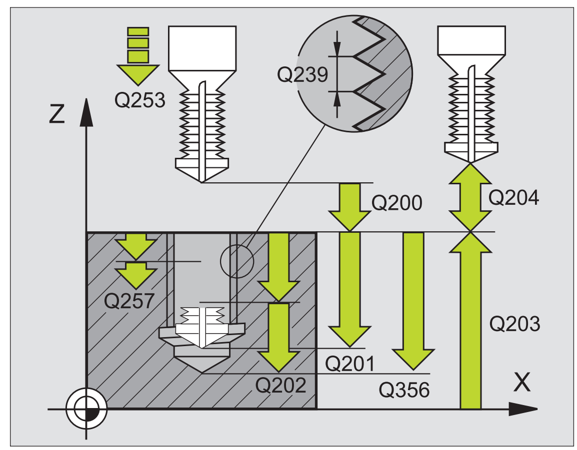
The algebraic sign of the cycle parameters depth of thread, countersinking depth or sinking depth at front determines the working direction. The working direction is defined in the following sequence:
1st: Depth of thread
2nd: Total hole depth
3rd: Depth at front

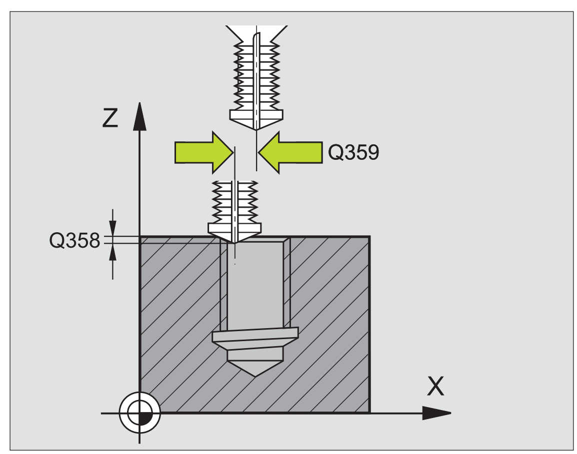
If you program a depth parameter to be 0, the TNC does not execute that step.

Program the thread depth as a value smaller than the total hole depth by at least one-third the thread pitch.

- **`Nominal diameter`** Q335: Nominal thread diameter.
- **`Thread pitch`** Q239: Pitch of the thread. The algebraic sign differentiates between right-hand and left-hand threads:
  + = right-hand thread
  – = left-hand thread
- **`Thread depth`** Q201 (incremental value): Distance between workpiece surface and root of thread.
- **`Total hole depth`** Q356 (incremental value): Distance between workpiece surface and bottom of hole.
- **`Feed rate for pre-positioning`** Q253: Traversing speed of the tool when moving in and out of the workpiece, in mm/min.
- **`Climb or up-cut`** Q351: Type of milling operation with M03
  +1 = climb milling
  –1 = up-cut milling
- **`Plunging depth`** Q202 (incremental value): Infeed per cut. The depth does not have to be a multiple of the plunging depth. The TNC will go to depth in one movement if:
  - the plunging depth is equal to the depth
  - the plunging depth is greater than the depth
- **`Upper advanced stop distance`** Q258 (incremental value): Set-up clearance for rapid traverse positioning when the TNC moves the tool again to the current plunging depth after retraction from the hole
- **`Infeed depth for chip breaking`** Q257 (incremental value): Depth at which TNC carries out chip breaking. There is no chip breaking if 0 is entered.
- **`Retraction rate for chip breaking`** Q256 (incremental value): Value by which the TNC retracts the tool during chip breaking.
- **`Depth at front`** Q358 (incremental value): Distance between tool point and the top surface of the workpiece for countersinking at the front of the tool.
- **`Countersinking offset at front`** Q359 (incremental value): Distance by which the TNC moves the tool center away from the hole center.







- **`Set-up clearance`** Q200 (incremental value): Distance between tool tip and workpiece surface.
- **`Workpiece surface coordinate`** Q203 (absolute value): Coordinate of the workpiece surface.
- **`2nd set-up clearance`** Q204 (incremental value): Coordinate in the tool axis at which no collision between tool and workpiece (clamping devices) can occur.
- **`Feed rate for plunging`** Q206: Traversing speed of the tool during drilling in mm/min.
- **`Feed rate for milling`** Q207: Traversing speed of the tool in mm/min while milling.

Example: NC block

```
N25 G264 Q335=10 Q239=+1.5 Q201=-16
    Q356=-20 Q253=750 Q351=+1 Q202=5
    Q258=0.2 Q257=5 Q256=0.2 Q358=+0
    Q359=+0 Q200=2 Q203=+30 Q204=50
    Q206=150 Q207=500 *
```

HELICAL THREAD DRILLING/MILLING (Cycle G265, not TNC 410)

1 The TNC positions the tool in the tool axis at rapid traverse to the programmed setup clearance above the workpiece surface.

Countersinking at front

2 If countersinking is before thread milling, the tool moves at the feed rate for countersinking to the sinking depth at front. If countersinking is after thread milling, the tool moves at the feed rate for pre-positioning to the countersinking depth.

3 The TNC positions the tool without compensation from the center on a semicircle to the offset at front, and then follows a circular path at the feed rate for countersinking.

4 The tool then moves in a semicircle to the hole center.

Thread milling

5 The tool moves at the programmed feed rate for pre-positioning to the starting plane for the thread.

6 The tool then approaches the thread diameter tangentially in a helical movement.

7 The tool moves on a continuous helical downward path until it reaches the thread depth.

8 After this, the tool departs the contour tangentially and returns to the starting point in the working plane.

9 At the end of the cycle, the TNC retracts the tool in rapid traverse to set-up clearance, or—if programmed—to the 2nd set-up clearance.

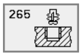
Before programming, note the following:

Program a positioning block for the starting point (hole center) in the working plane with radius compensation **G40**.

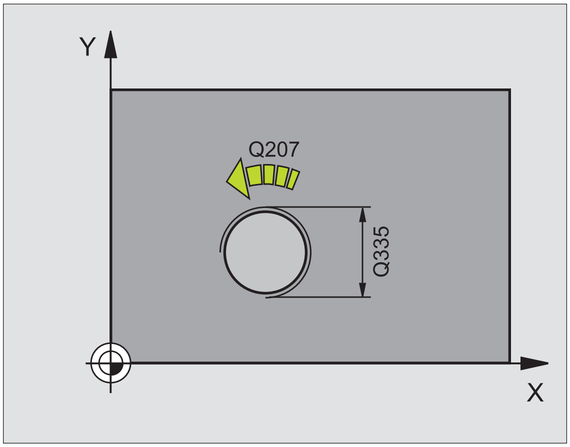
The algebraic sign of the cycle parameters depth of thread or sinking depth at front determines the working direction. The working direction is defined in the following sequence:
1st: Depth of thread
2nd: Depth at front

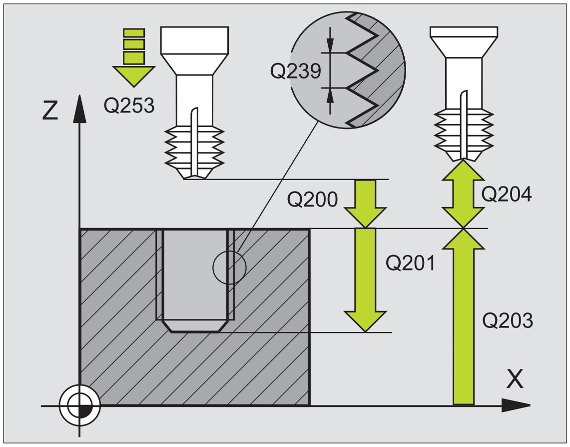
If you program a depth parameter to be 0, the TNC does not execute that step.

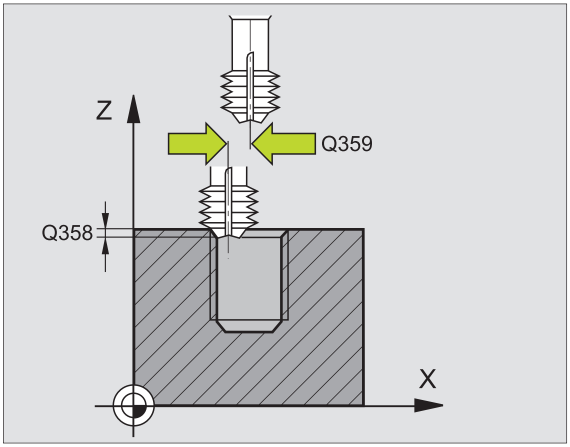
The type of milling (up-cut/climb) is determined by the thread (right-hand/left-hand) and the direction of tool rotation, since it is only possible to work in the direction of the tool.



- **`Nominal diameter`** Q335: Nominal thread diameter.
- **`Thread pitch`** Q239: Pitch of the thread. The algebraic sign differentiates between right-hand and left-hand threads:
  + = right-hand thread
  – = left-hand thread
- **`Thread depth`** Q201 (incremental value): Distance between workpiece surface and root of thread.
- **`Feed rate for pre-positioning`** Q253: Traversing speed of the tool when moving in and out of the workpiece, in mm/min.
- **`Depth at front`** Q358 (incremental value): Distance between tool point and the top surface of the workpiece for countersinking at the front of the tool.
- **`Countersinking offset at front`** Q359 (incremental value): Distance by which the TNC moves the tool center away from the hole center.
- **`Countersink`** Q360: Execution of the chamfer
  0 = before thread machining
  1 = after thread machining
- **`Set-up clearance`** Q200 (incremental value): Distance between tool tip and workpiece surface.







- **Workpiece surface coordinate** Q203 (absolute value): Coordinate of the workpiece surface.
- **2nd set-up clearance** Q204 (incremental value): Coordinate in the tool axis at which no collision between tool and workpiece (clamping devices) can occur.
- **Feed rate for counterboring** Q254: Traversing speed of the tool during counterboring in mm/min.
- **Feed rate for milling** Q207: Traversing speed of the tool in mm/min while milling.

Example: NC block

```
N25 G265 Q335=10 Q239=+1.5 Q201=-16
    Q253=750 Q358=+0 Q359=+0
    Q360=0 Q200=2 Q203=+30 Q204=50
    Q254=150 Q207=500 *
```

OUTSIDE THREAD MILLING (Cycle G267, not TNC 410)

1 The TNC positions the tool in the tool axis at rapid traverse to the programmed setup clearance above the workpiece surface.

Countersinking at front

2 The TNC moves on the reference axis of the working plane from the center of the stud to the starting point for countersinking at front. The position of the starting point is determined by the thread radius, tool radius and pitch.
3 The tool moves at the feed rate for pre-positioning to the sinking depth at front.
4 The TNC positions the tool without compensation from the center on a semicircle to the offset at front, and then follows a circular path at the feed rate for countersinking.
5 The tool then moves in a semicircle to the starting point.

Thread milling

6 The TNC positions the tool to the starting point if there has been no previous countersinking at front. Starting point for thread milling = starting point for countersinking at front.
7 The tool moves at the programmed feed rate for pre-positioning to the starting plane. The starting plane is derived from the algebraic sign of the thread pitch, the milling method (climb or up-cut milling) and the number of threads per step.
8 The tool then approaches the thread diameter tangentially in a helical movement.
9 Depending on the setting of the parameter for the number of threads, the tool mills the thread in one, in several spaced or in one continuous helical movement.
10 After this, the tool departs the contour tangentially and returns to the starting point in the working plane.

11 At the end of the cycle, the TNC retracts the tool in rapid traverse to set-up clearance, or—if programmed—to the 2nd set-up clearance.

Before programming, note the following:

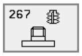
Program a positioning block for the starting point (stud center) in the working plane with radius compensation **G40**.

The offset required before countersinking at the front should be determined ahead of time. You must enter the value from the center of the stud to the center of the tool (uncorrected value).

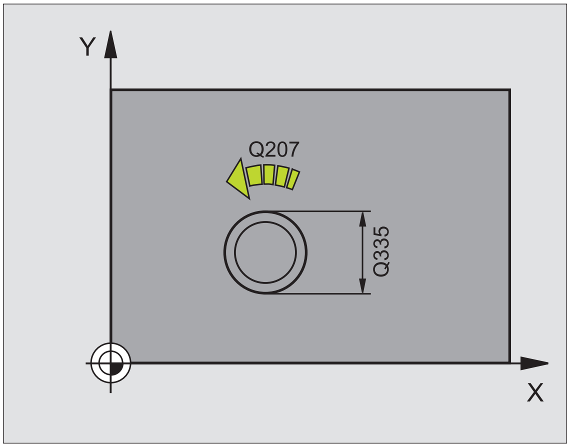
The algebraic sign of the cycle parameters depth of thread, countersinking depth or sinking depth at front determines the working direction. The working direction is defined in the following sequence:
1st: Depth of thread
2nd: Depth at front

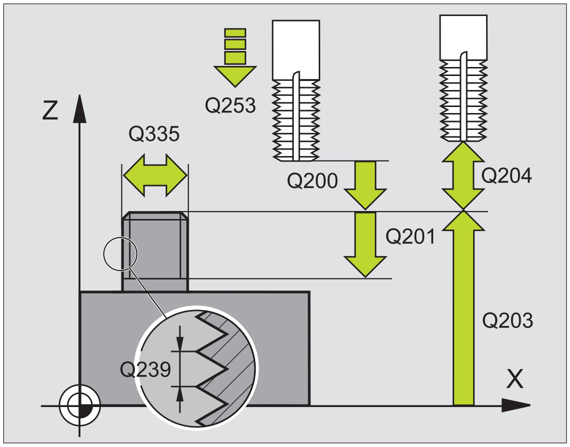
If you program a depth parameter to be 0, the TNC does not execute that step.

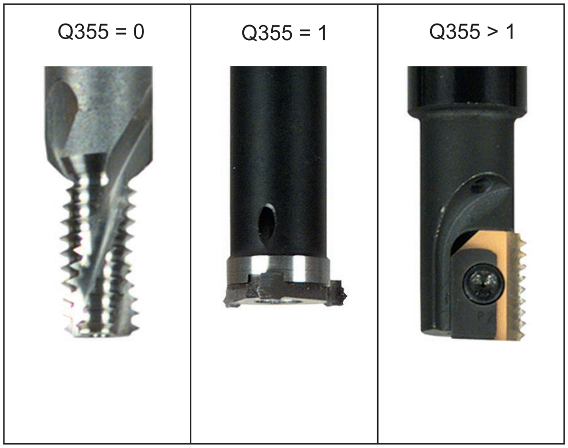
The algebraic sign for the cycle parameter thread depth determines the working direction.



- **Nominal diameter** Q335: Nominal thread diameter.
- **Thread pitch** Q239: Pitch of the thread. The algebraic sign differentiates between right-hand and left-hand threads:
  + = right-hand thread
  – = left-hand thread
- **Thread depth** Q201 (incremental value): Distance between workpiece surface and root of thread.
- **Threads per step** Q355: Number of thread revolutions by which the tool is offset, see figure at lower right
  0 = one 360° helical path to the depth of thread
  1 = continuous helical path over the entire length of the thread
  >1 = several helical paths with approach and departure; between them, the TNC offsets the tool by Q355, multiplied by the pitch
- **Feed rate for pre-positioning** Q253: Traversing speed of the tool when moving in and out of the workpiece, in mm/min.
- **Climb or up-cut** Q351: Type of milling operation with M03
  +1 = climb milling
  –1 = up-cut milling





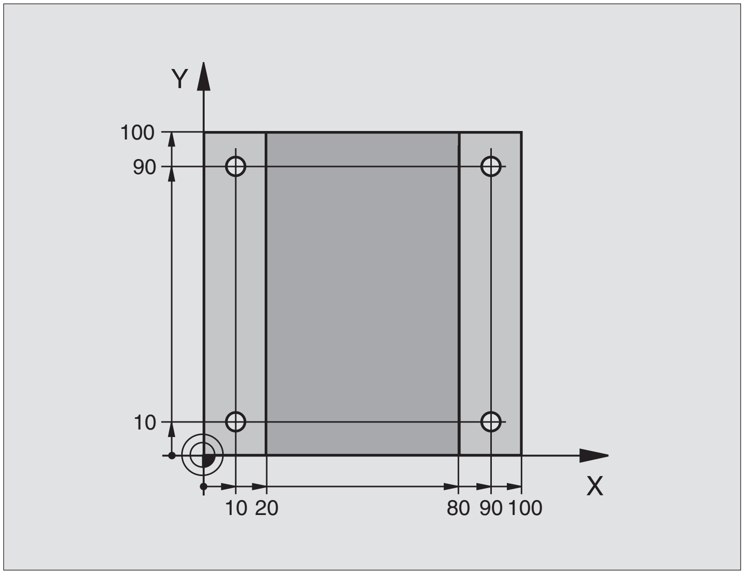


- **Set-up clearance** Q200 (incremental value): Distance between tool tip and workpiece surface.
- **Depth at front** Q358 (incremental value): Distance between tool point and the top surface of the workpiece for countersinking at the front of the tool.
- **Countersinking offset at front** Q359 (incremental value): Distance by which the TNC moves the tool center away from the stud center.
- **Workpiece surface coordinate** Q203 (absolute value): Coordinate of the workpiece surface.
- **2nd set-up clearance** Q204 (incremental value): Coordinate in the tool axis at which no collision between tool and workpiece (clamping devices) can occur.
- **Feed rate for counterboring** Q254: Traversing speed of the tool during counterboring in mm/min.
- **Feed rate for milling** Q207: Traversing speed of the tool in mm/min while milling.

Example: NC block

```
N25 G267 Q335=10 Q239=+1.5 Q201=-20
    Q355=0 Q253=750 Q351=+1 Q200=2
    Q358=+0 Q359=+0 Q203=+30 Q204=50
    Q254=150 Q207=500 *
```

Example: Drilling cycles

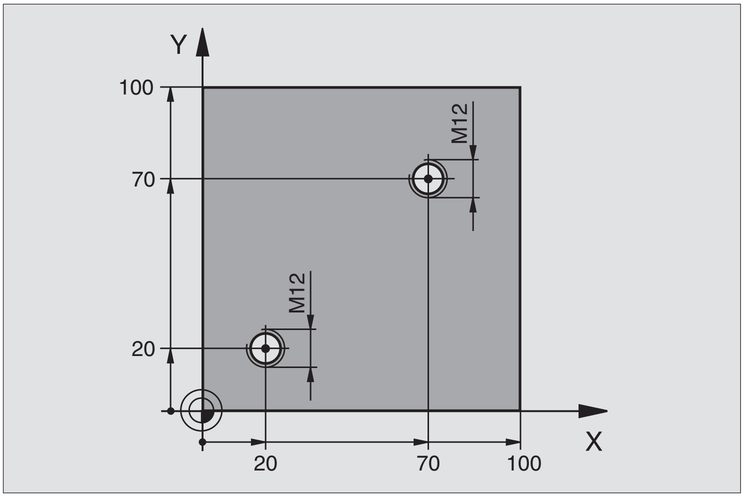


%C200 G71 *	
N10 G30 G17 X+0 Y+0 Z-20 *	Define the workpiece blank
N20 G31 G90 X+100 Y+100 Z+0 *	
N30 G99 T1 L+0 R+3 *	Define the tool
N40 T1 G17 S4500 *	Tool call
N50 G00 G40 G90 Z+250 *	Retract the tool
N60 G200 Q200=2 Q201=-15 Q206=250	Define cycle
Q202=5 Q210=0 Q203=0 Q204=50 *	
N70 X+10 Y+10 M3 *	Approach hole 1, spindle ON
N80 Z-8 M99 *	Pre-position in the spindle axis, cycle call
N90 Y+90 M99 *	Approach hole 2, call cycle
N100 Z+20 *	Retract in the spindle axis
N110 X+90 *	Approach hole 3
N120 Z-8 M99 *	Pre-position in the spindle axis, cycle call
N130 Y+10 M99 *	Approach hole 4, call cycle
N140 G00 Z+250 M2 *	Retract in the tool axis, end program
N999999 %C200 G71 *	Call the cycle

Example: Drilling cycles

Program sequence

- Program the drilling cycle in the main program
- Program machining within a subprogram, see “Subprograms,” page 317



%C18 G71 *	
N10 G30 G17 X+0 Y+0 Z-20 *	Define the workpiece blank
N20 G31 G90 X+100 Y+100 Z+0 *	
N30 G99 T1 L+0 R+6 *	Define the tool
N40 T1 G17 S4500 *	Tool call
N50 G00 G40 G90 Z+250 *	Retract the tool
N60 G86 P01 +30 P02 -1.75 *	Define THREAD CUTTING cycle
N70 X+20 Y+20 *	Approach hole 1
N80 L1.0 *	Call subprogram 1
N90 X+70 Y+70 *	Approach hole 2
N100 L1.0 *	Call subprogram 1
N110 G00 Z+250 M2 *	Retract tool, end of main program
N120 G98 L1 *	Subprogram 1: Thread cutting
N130 G36 S0 *	Define angle of spindle orientation
N140 M19 *	Orient spindle (makes it possible to cut repeatedly)
N150 G01 G91 X-2 F1000 *	Tool offset to prevent collision during tool infeed (dependent
	on core diameter and tool)
N160 G90 Z-30 *	Move to starting depth
N170 G91 X+2 *	Reset the tool to hole center
N180 G79 *	Call Cycle 18
N190 G90 Z+5 *	Retract tool
N200 G98 L0 *	End of subprogram 1
N999999 %C18 G71 *	

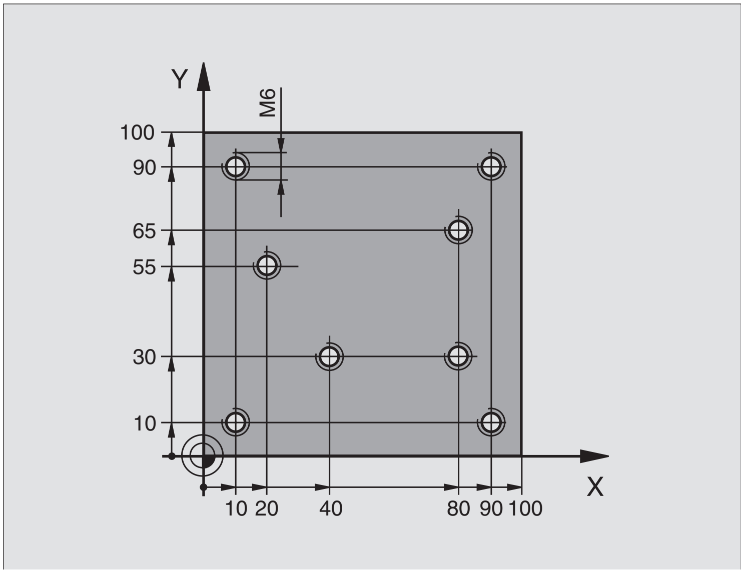
Example: Calling drilling cycles in connection with a point table (only with TNC 410)

The drill hole coordinates are stored in the point table TAB1.PNT and are called by the TNC with G79 PAT.

The tool radii are selected so that all work steps can be seen in the test graphics.

Program sequence

- Centering
- Drilling
- Tapping



`%1 G71*`	
`N10 G30 G17 X+0 Y+0 Z-20 *`	Define the workpiece blank
`N20 G31 X+100  Y+100  Z+0 *`	
`N30 G99 1 L+0 R+4 *`	Tool definition of center drill
`N40 G99 2 L+0 R+2.4 *`	Define tool: drill
`N50 G99 3 L+0 R+3 *`	Tool definition of tap
`N60 T1 G17 S5000 *`	Tool call of centering drill
`N70 G01 G40 Z+10 F5000 *`	Move tool to clearance height (Enter a value for F.
	The TNC positions to the clearance height after every cycle)
`N80 %:PAT: "TAB1" *`	Defining point tables
`N90 G200 Q200=2 Q201=-2 Q206=150 Q202=2`	Cycle definition: Centering
`    Q210=0 Q203=+0 Q204=0*`	The value 0 must be entered with Q203 and Q204.
`N100 G79 "PAT" F5000 M3 *`	Cycle call in connection with point table TAB1.PNT
	Feed rate between points: 5000 mm/min
`N110 G00 G40 Z+100 M6 *`	Retract the tool, change the tool
`N120 T2 G17 S5000 *`	Call the drilling tool
`N130 G01 G40 Z+10 F5000 *`	Move tool to clearance height (enter a value for F)
`N140 G200 Q200=2 Q201=-25 Q206=150 Q202=5`	Cycle definition: drilling
`     Q210=0 Q203=+0 Q204=0*`	The value 0 must be entered with Q203 and Q204.
`N150 G79 "PAT" F5000 M3 *`	Cycle call in connection with point table TAB1.PNT

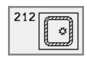
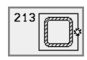
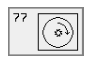
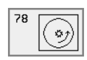
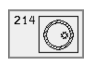
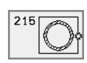
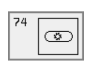
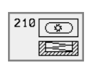
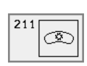
`N160 G00 G40 Z+100 M6 *`	Retract the tool, change the tool
`N170 T3 G17 S200 *`	Tool call for tap
`N180 G00 G40 Z+50 *`	Move tool to clearance height
`N190 G84 P01 +2 P02 -15 P030 P04 150 *`	Cycle definition for tapping
`N200 G79 "PAT" F5000 M3 *`	Cycle call in connection with point table TAB1.PNT
`N210  G00 G40 Z+100 M2*`	Retract in the tool axis, end program
`N99999 %1 G71*`	

Point table TAB1.PNT

```
    TAB1. PNT   MM
NR    X      Y      Z
0     +10    +10    +0
1     +40    +30    +0
2     +90    +10    +0
3     +80    +30    +0
4     +80    +65    +0
5     +90    +90    +0
6     +10    +90    +0
7     +20    +55    +0
[END]
```

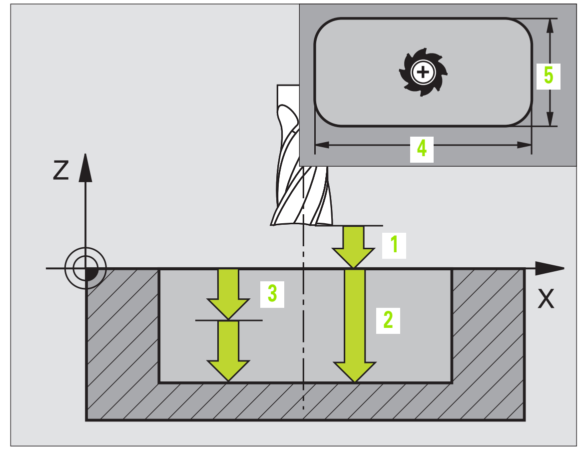
8.4 Cycles for Milling Pockets, Studs and Slots

Overview

Cycle	Soft key
G75/G76 POCKET MILLING (rectangular) Roughing cycle without automatic pre-positioning G75: In clockwise direction G76: In counterclockwise direction	 
G212 POCKET FINISHING (rectangular) Finishing cycle with automatic pre-positioning, 2nd set-up clearance	
G213 STUD FINISHING (rectangular) Finishing cycle with automatic pre-positioning, 2nd set-up clearance	
G77/G78 CIRCULAR POCKET MILLING Roughing cycle without automatic pre-positioning G77: In clockwise direction G78: In counterclockwise direction	 
G214 CIRCULAR POCKET FINISHING Finishing cycle with automatic pre-positioning, 2nd set-up clearance	
G215 CIRCULAR STUD FINISHING Finishing cycle with automatic pre-positioning, 2nd set-up clearance	
G74 SLOT MILLING Roughing/finishing cycle without automatic pre-positioning, vertical depth infeed	
G210 SLOT RECIP. PLNG Roughing/finishing cycle with automatic pre-positioning, with reciprocating plunge infeed	
G211 CIRCULAR SLOT Roughing/finishing cycle with automatic pre-positioning, with reciprocating plunge infeed	

POCKET MILLING (Cycles G75, G76)

1 The tool penetrates the workpiece at the starting position (pocket center) and advances to the first plunging depth.
2 The cutter begins milling in the positive axis direction of the longer side (on square pockets, always starting in the positive Y direction) and then roughs out the pocket from the inside out.
3 This process (1 to 2) is repeated until the depth is reached.
4 At the end of the cycle, the TNC retracts the tool to the starting position.



Before programming, note the following:

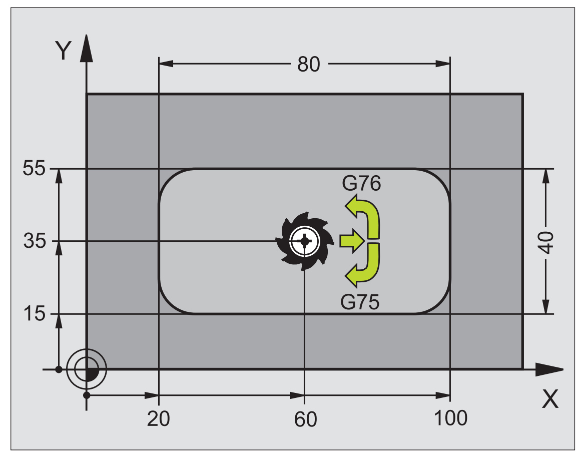
This cycle requires a center-cut end mill (ISO 1641), or pilot drilling at the pocket center.

Pre-position over the pocket center with radius compensation **G40.**

Program a positioning block for the starting point in the tool axis (set-up clearance above the workpiece surface).

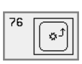
The algebraic sign for the cycle parameter DEPTH determines the working direction. If you program DEPTH = 0, the cycle will not be executed.

The following prerequisite applies for the 2nd side length: 2nd side length greater than [(2 x rounding radius) + stepover factor k].



Direction of rotation during rough-out

- In clockwise direction: G75 (DR-)
- In counterclockwise direction: G76 (DR+)



- **Set-up clearance 1** (incremental value): Distance between tool tip (at starting position) and workpiece surface
- **Milling depth 2** (incremental value): Distance between workpiece surface and bottom of pocket
- **Plunging depth 3** (incremental value): Infeed per cut The TNC will go to depth in one movement if:
  - the plunging depth is equal to the depth
  - the plunging depth is greater than the depth
- **Feed rate for plunging:** Traversing speed of the tool during penetration

Example: NC blocks

```
N27 G75 P01 2  P02 -20  P03 5  P04 100
    P05 X+80  P06 Y+40  P07 275 P08 5 *
...
N35 G76 P01 2  P02 -20  P03 5  P04 100
    P05 X+80  P06 Y+40  P07 275 P08 5 *
```

- **First side length 4** (incremental value): Pocket length, parallel to the reference axis of the working plane
- **2nd side length 5**: Pocket width
- **Feed rate F**: Traversing speed of the tool in the working plane
- **Rounding off radius**: Radius for the pocket corners. If Radius = 0 is entered, the pocket corners will be rounded with the radius of the cutter.

Calculations:
Stepover factor k = K x R

K: is the overlap factor, preset in Machine Parameter 7430, and
R is the cutter radius

POCKET FINISHING (Cycle G212)

1 The TNC automatically moves the tool in the tool axis to set-up clearance, or—if programmed—to the 2nd set-up clearance, and subsequently to the center of the pocket.

2 From the pocket center, the tool moves in the working plane to the starting point for machining. The TNC takes the allowance and tool radius into account for calculating the starting point. If necessary, the TNC penetrates at the pocket center.

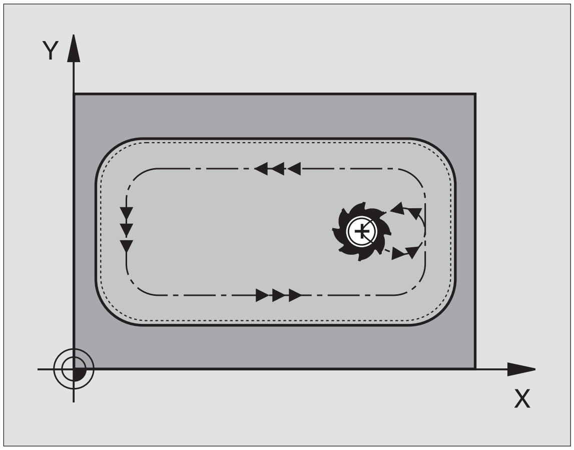
3 If the tool is at the 2nd set-up clearance, it moves at rapid traverse to the set-up clearance, and from there advances to the first plunging depth at the feed rate for plunging.

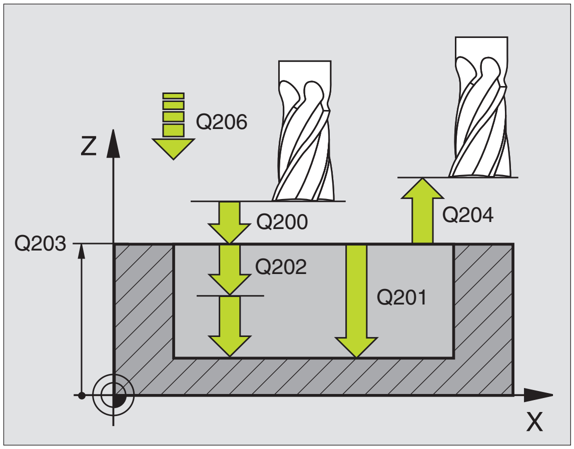
4 The tool then moves tangentially to the contour of the finished part and, using climb milling, machines one revolution.

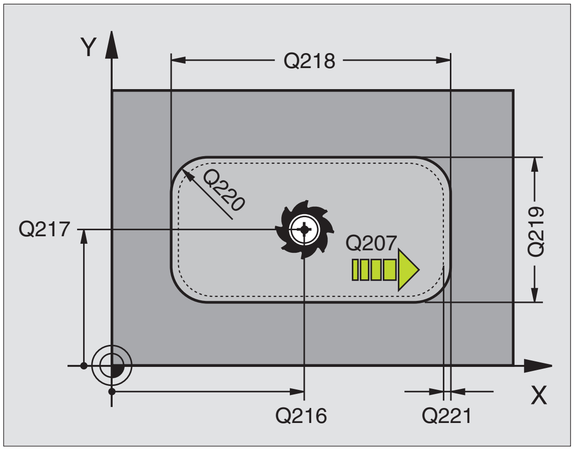
5 The tool then departs the contour on a tangential path and returns to the starting point in the working plane.

6 This process (3 to 5) is repeated until the programmed depth is reached.

7 At the end of the cycle, the TNC retracts the tool in rapid traverse to set-up clearance, or—if programmed—to the 2nd set-up clearance, and finally to the center of the pocket (end position = starting position).







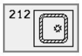
Before programming, note the following:

The TNC automatically pre-positions the tool in the tool axis and working plane.

The algebraic sign for the cycle parameter DEPTH determines the working direction. If you program DEPTH = 0, the cycle will not be executed.

If you want to clear and finish the pocket with the same tool, use a center-cut end mill (ISO 1641) and enter a low feed rate for plunging.

Minimum size of the pocket: 3 times the tool radius.

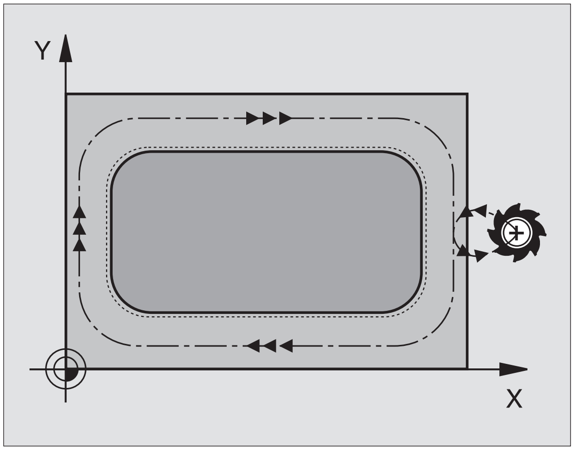


- **Set-up clearance** Q200 (incremental value): Distance between tool tip and workpiece surface.
- **Depth** Q201 (incremental value): Distance between workpiece surface and bottom of pocket.
- **Feed rate for plunging** Q206: Traversing speed of the tool in mm/min when moving to depth. If you are plunge-cutting into the material, enter a value lower than that defined in Q207.
- **Plunging depth** Q202 (incremental value): Infeed per cut. Enter a value greater than 0.
- **Feed rate for milling** Q207: Traversing speed of the tool in mm/min while milling.
- **Workpiece surface coordinate** Q203 (absolute value): Coordinate of the workpiece surface.
- **2nd set-up clearance** Q204 (incremental value): Coordinate in the tool axis at which no collision between tool and workpiece (clamping devices) can occur.
- **Center in 1st axis** Q216 (absolute value): Center of the pocket in the reference axis of the working plane.
- **Center in 2nd axis** Q217 (absolute value): Center of the pocket in the minor axis of the working plane.
- **First side length** Q218 (incremental value): Pocket length, parallel to the reference axis of the working plane.
- **Second side length** Q219 (incremental value): Pocket length, parallel to the minor axis of the working plane
- **Corner radius** Q220: Radius of the pocket corner: If you make no entry here, the TNC assumes that the corner radius is equal to the tool radius.
- **Allowance in 1st axis** Q221 (incremental value): Allowance for pre-positioning in the reference axis of the working plane referenced to the length of the pocket.

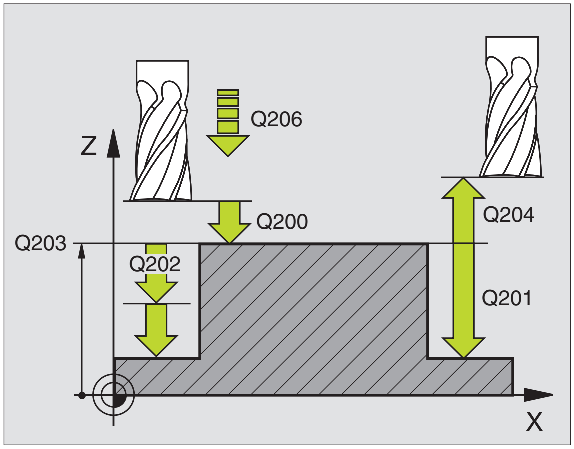
Example: NC block

```
N34 G212 Q200=2 Q201=-20 Q206=150 Q202=5
    Q207=500 Q203=+30 Q204=50 Q216=+50
    Q217=+50 Q218=80 Q219=60 Q220=5
    Q221=0 *
```

STUD FINISHING (Cycle G213)



1 The TNC moves the tool in the tool axis to set-up clearance, or—if programmed—to the 2nd set-up clearance, and subsequently to the center of the stud.
2 From the stud center, the tool moves in the working plane to the starting point for machining. The starting point lies to the right of the stud by a distance approx. 3.5 times the tool radius.
3 If the tool is at the 2nd set-up clearance, it moves in rapid traverse to set-up clearance, and from there advances to the first plunging depth at the feed rate for plunging.
4 The tool then moves tangentially to the contour of the finished part and, using climb milling, machines one revolution.
5 The tool then departs the contour on a tangential path and returns to the starting point in the working plane.
6 This process (3 to 5) is repeated until the programmed depth is reached.
7 At the end of the cycle, the TNC retracts the tool at rapid traverse to the set-up clearance, or—if programmed—to the 2nd set-up clearance, and finally to the center of the stud (end position = starting position).

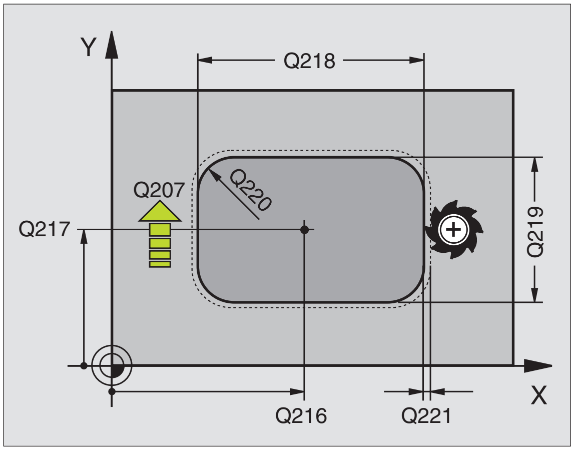


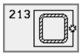
Before programming, note the following:

The TNC automatically pre-positions the tool in the tool axis and working plane.

The algebraic sign for the cycle parameter DEPTH determines the working direction. If you program DEPTH = 0, the cycle will not be executed.

If you want to clear and finish the stud with the same tool, use a center-cut end mill (ISO 1641) and enter a low feed rate for plunging.



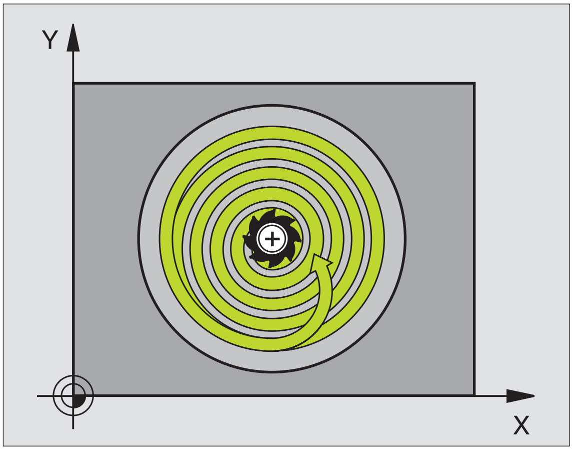


- **Set-up clearance** Q200 (incremental value): Distance between tool tip and workpiece surface.
- **Depth** Q201 (incremental value): Distance between workpiece surface and bottom of stud.
- **Feed rate for plunging** Q206: Traversing speed of the tool in mm/min when moving to depth. If you are plunge-cutting into the material, enter a low value; if you have already cleared the stud, enter a higher feed rate.
- **Plunging depth** Q202 (incremental value): Infeed per cut. Enter a value greater than 0.
- **Feed rate for milling** Q207: Traversing speed of the tool in mm/min while milling.
- **Workpiece surface coordinate** Q203 (absolute value): Coordinate of the workpiece surface.
- **2nd set-up clearance** Q204 (incremental value): Coordinate in the tool axis at which no collision between tool and workpiece (clamping devices) can occur.
- **Center in 1st axis** Q216 (absolute value): Center of the stud in the reference axis of the working plane.
- **Center in 2nd axis** Q217 (absolute value): Center of the stud in the minor axis of the working plane.
- **First side length** Q218 (incremental value): Length of stud parallel to the reference axis of the working plane
- **Second side length** Q219 (incremental value): Length of stud parallel to the secondary axis of the working plane
- **Corner radius** Q220: Radius of the stud corner.
- **Allowance in 1st axis** Q221 (incremental value): Allowance for pre-positioning in the reference axis of the working plane referenced to the length of the stud.

Example: NC block

```
N35 G213 Q200=2 Q201=-20 Q206=150 Q202=5
    Q207=500 Q203=+30 Q204=50 Q216=+50
    Q217=+50 Q218=80 Q219=60 Q220=5
    Q221=0 *
```

CIRCULAR POCKET MILLING (Cycle G77, G78)



1 The tool penetrates the workpiece at the starting position (pocket center) and advances to the first plunging depth.

2 The tool subsequently follows a spiral path at the feed rate F - see figure at right. For calculating the stepover factor k, see Cycle 4 POCKET MILLING.see "POCKET MILLING (Cycles G75, G76)," page 232

3 This process is repeated until the depth is reached.

4 At the end of the cycle, the TNC retracts the tool to the starting position.

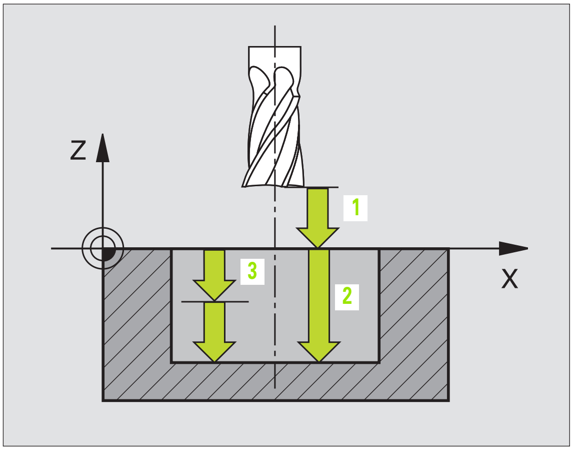
Before programming, note the following:

This cycle requires a center-cut end mill (ISO 1641), or pilot drilling at the pocket center.

Pre-position over the pocket center with radius compensation **G40.**

Program a positioning block for the starting point in the tool axis (set-up clearance above the workpiece surface).

The algebraic sign for the cycle parameter DEPTH determines the working direction. If you program DEPTH = 0, the cycle will not be executed.



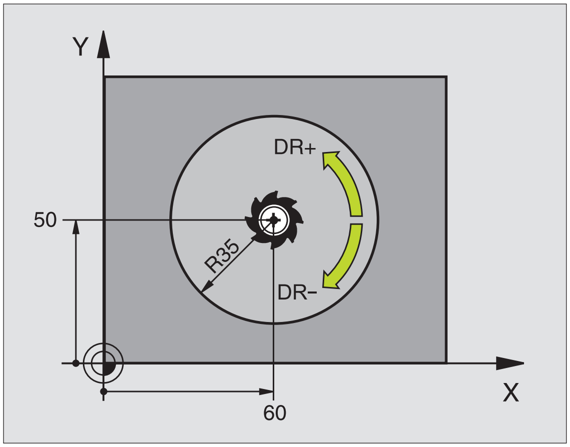
Direction of rotation during rough-out

- In clockwise direction: G77 (DR-)
- In counterclockwise direction: G78 (DR+)

78

- **Set-up clearance 1** (incremental value): Distance between tool tip (at starting position) and workpiece surface
- **Milling depth 2**: Distance between workpiece surface and bottom of pocket
- **Plunging depth 3** (incremental value): Infeed per cut The TNC will go to depth in one movement if:
  - the plunging depth is equal to the depth
  - the plunging depth is greater than the depth

- **Feed rate for plunging:** Traversing speed of the tool during penetration
- **Circular radius**: Radius of the circular pocket
- **Feed rate F**: Traversing speed of the tool in the working plane



Example: NC blocks

```
N26 G77 P01 2  P02 -20  P035  P04 100
    P05 40  P06 250 *
...
N48 G78 P01 2  P02 -20  P03 5  P04 100
    P05 40  P06 250 *
```

CIRCULAR POCKET FINISHING (Cycle G214)

1 The TNC automatically moves the tool in the tool axis to set-up clearance, or—if programmed—to the 2nd set-up clearance, and subsequently to the center of the pocket.

2 From the pocket center, the tool moves in the working plane to the starting point for machining. The TNC takes the workpiece blank diameter and tool radius into account for calculating the starting point. If you enter a workpiece blank diameter of 0, the TNC plunge-cuts into the pocket center.

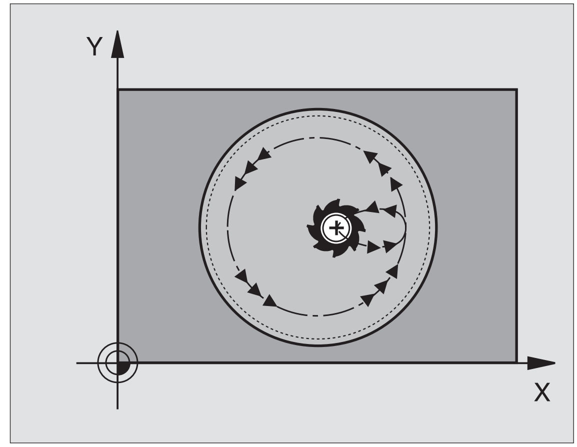
3 If the tool is at the 2nd set-up clearance, it moves in rapid traverse to set-up clearance, and from there advances to the first plunging depth at the feed rate for plunging.

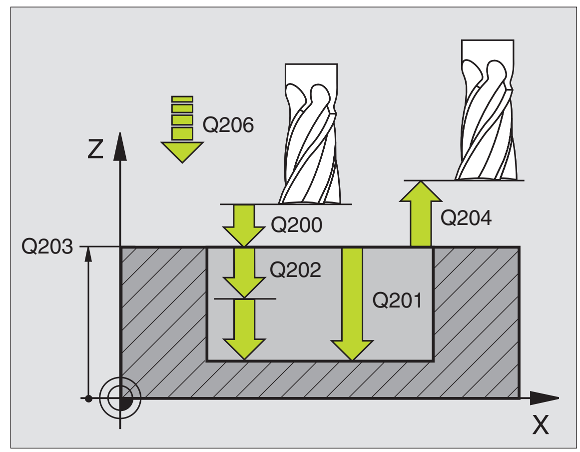
4 The tool then moves tangentially to the contour of the finished part and, using climb milling, machines one revolution.

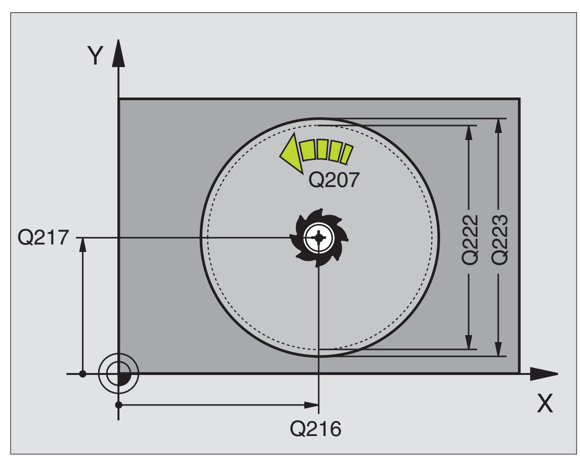
5 After this, the tool departs the contour tangentially and returns to the starting point in the working plane.

6 This process (3 to 5) is repeated until the programmed depth is reached.

7 At the end of the cycle, the TNC retracts the tool at rapid traverse to set-up clearance, or, if programmed, to the 2nd set-up clearance and then to the center of the pocket (end position = starting position)





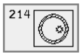


Before programming, note the following:

The TNC automatically pre-positions the tool in the tool axis and working plane.

The algebraic sign for the cycle parameter DEPTH determines the working direction. If you program DEPTH = 0, the cycle will not be executed.

If you want to clear and finish the pocket with the same tool, use a center-cut end mill (ISO 1641) and enter a low feed rate for plunging.



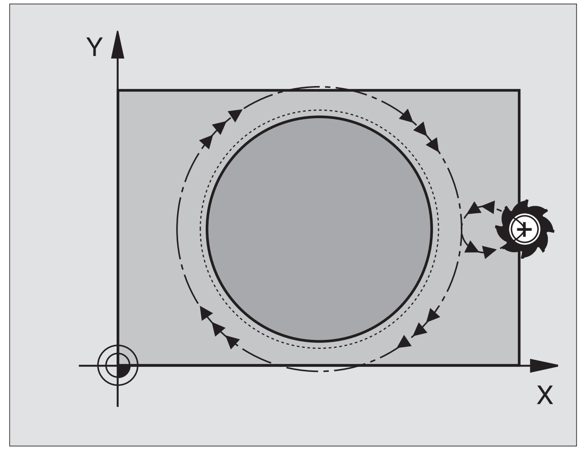
- **`Set-up clearance`** Q200 (incremental value): Distance between tool tip and workpiece surface.
- **`Depth`** Q201 (incremental value): Distance between workpiece surface and bottom of pocket.
- **`Feed rate for plunging`** Q206: Traversing speed of the tool in mm/min when moving to depth. If you are plunge-cutting into the material, enter a value lower than that defined in Q207.
- **`Plunging depth`** Q202 (incremental value): Infeed per cut.
- **`Feed rate for milling`** Q207: Traversing speed of the tool in mm/min while milling.
- **`Workpiece surface coordinate`** Q203 (absolute value): Coordinate of the workpiece surface.
- **`2nd set-up clearance`** Q204 (incremental value): Coordinate in the tool axis at which no collision between tool and workpiece (clamping devices) can occur.
- **`Center in 1st axis`** Q216 (absolute value): Center of the pocket in the reference axis of the working plane.
- **`Center in 2nd axis`** Q217 (absolute value): Center of the pocket in the minor axis of the working plane.
- **`Workpiece blank diameter`** Q222: Diameter of the premachined pocket for calculating the pre-position. Enter the workpiece blank diameter to be less than the diameter of the finished part.
- **`Finished part diameter`** Q223: Diameter of the finished pocket. Enter the diameter of the finished part to be greater than the workpiece blank diameter.

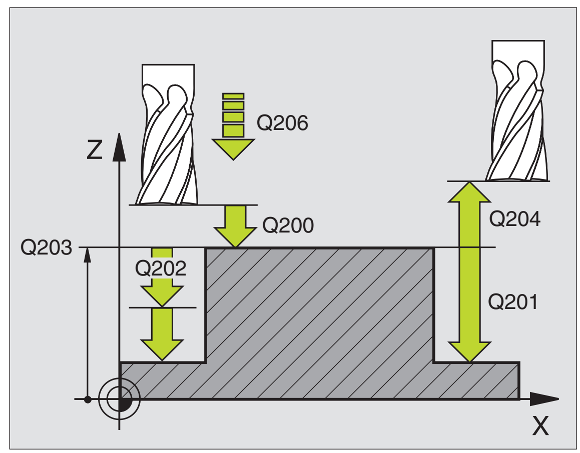
Example: NC block

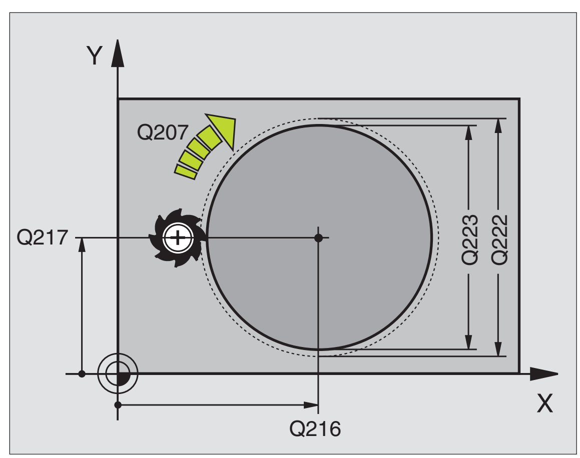
```
N42 G214 Q200=2 Q201=-20 Q206=150 Q202=5
    Q207=500 Q203=+30 Q204=50 Q216=+50
    Q217=+50 Q222=79 Q223=80 *
```

CIRCULAR STUD FINISHING (Cycle G215)

1 The TNC automatically moves the tool in the tool axis to set-up clearance, or—if programmed—to the 2nd set-up clearance, and subsequently to the center of the stud.
2 From the stud center, the tool moves in the working plane to the starting point for machining. The starting point lies to the right of the stud by a distance approx. 3.5 times the tool radius.
3 If the tool is at the 2nd set-up clearance, it moves in rapid traverse to set-up clearance, and from there advances to the first plunging depth at the feed rate for plunging.
4 The tool then moves tangentially to the contour of the finished part and, using climb milling, machines one revolution.
5 The tool then departs the contour on a tangential path and returns to the starting point in the working plane.
6 This process (3 to 5) is repeated until the programmed depth is reached.
7 At the end of the cycle, the TNC retracts the tool at rapid traverse to the set-up clearance, or—if programmed—to the 2nd set-up clearance, and finally to the center of the pocket (end position = starting position).





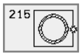


Before programming, note the following:

The TNC automatically pre-positions the tool in the tool axis and working plane.

The algebraic sign for the cycle parameter DEPTH determines the working direction. If you program DEPTH = 0, the cycle will not be executed.

If you want to clear and finish the stud with the same tool, use a center-cut end mill (ISO 1641) and enter a low feed rate for plunging.

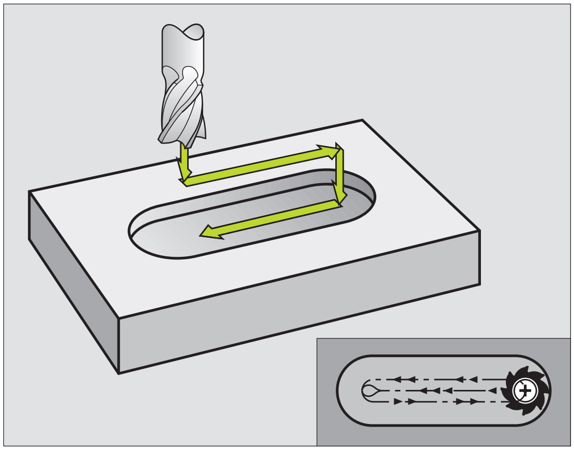


- **Set-up clearance** Q200 (incremental value): Distance between tool tip and workpiece surface.
- **Depth** Q201 (incremental value): Distance between workpiece surface and bottom of stud.
- **Feed rate for plunging** Q206: Traversing speed of the tool in mm/min when moving to depth. If you are plunge-cutting into the material, enter a low value; if you have already cleared the stud, enter a higher feed rate.
- **Plunging depth** Q202 (incremental value): Infeed per cut. Enter a value greater than 0.
- **Feed rate for milling** Q207: Traversing speed of the tool in mm/min while milling.
- **Workpiece surface coordinate** Q203 (absolute value): Coordinate of the workpiece surface.
- **2nd set-up clearance** Q204 (incremental value): Coordinate in the tool axis at which no collision between tool and workpiece (clamping devices) can occur.
- **Center in 1st axis** Q216 (absolute value): Center of the stud in the reference axis of the working plane.
- **Center in 2nd axis** Q217 (absolute value): Center of the stud in the minor axis of the working plane.
- **Workpiece blank diameter** Q222: Diameter of the premachined stud for calculating the pre-position. Enter the workpiece blank diameter to be greater than the diameter of the finished part.
- **Diameter of finished part** Q223: Diameter of the finished stud. Enter the diameter of the finished part to be less than the workpiece blank diameter.

Example: NC block

```
N43 G215 Q200=2 Q201=-20 Q206=150 Q202=5
    Q207=500 Q203=+30 Q204=50 Q216=+50
    Q217=+50 Q222=81 Q223=80 *
```

SLOT MILLING (Cycle G74)



Roughing process

1 The TNC moves the tool inward by the milling allowance (half the difference between the slot width and the tool diameter). From there it plunge-cuts into the workpiece and mills in the longitudinal direction of the slot.

2 After downfeed at the end of the slot, milling is performed in the opposite direction. This process is repeated until the programmed milling depth is reached.

Finishing process

3 The TNC advances the tool at the slot bottom on a tangential arc to the outside contour. The tool subsequently climb mills the contour (with M3).

4 At the end of the cycle, the tool is retracted at rapid traverse to the set-up clearance. If the number of infeeds was odd, the tool returns to the starting position at the level of the set-up clearance.

Before programming, note the following:

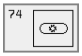
This cycle requires a center-cut end mill (ISO 1641), or pilot drilling at the starting point.

Pre-position to the center of the slot and offset by the tool radius into the slot with radius compensation **G40**.

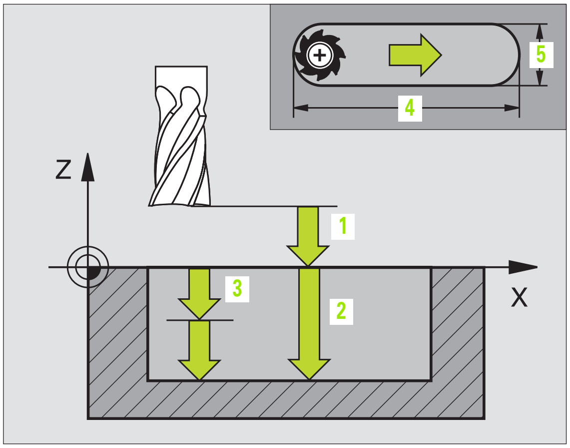
The cutter diameter must be not be larger than the slot width and not smaller than half the slot width.

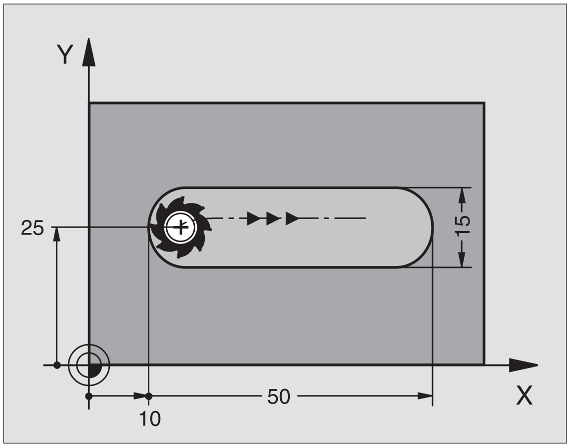
Program a positioning block for the starting point in the tool axis (set-up clearance above the workpiece surface).

The algebraic sign for the cycle parameter DEPTH determines the working direction. If you program DEPTH = 0, the cycle will not be executed.



- **Set-up clearance 1** (incremental value): Distance between tool tip (at starting position) and workpiece surface
- **Milling depth 2** (incremental value): Distance between workpiece surface and bottom of pocket
- **Plunging depth 3** (incremental value): Infeed per cut. The tool will drill to the depth in one operation if:
  - the plunging depth is equal to the depth
  - the plunging depth is greater than the depth
- **Feed rate for plunging**: Traversing speed during penetration
- **1st side length 4**: Slot length; specify the sign to determine the first milling direction
- **2nd side length 5**: Slot width
- **Feed rate F**: Traversing speed of the tool in the working plane





Example: NC block

```
N44 G74 P01 2  P02 -20  P0  5  P04 100
    P05 X+80  P06 Y+12 P07 275 *
```

SLOT with reciprocating plunge-cut (Cycle G210)

Before programming, note the following:

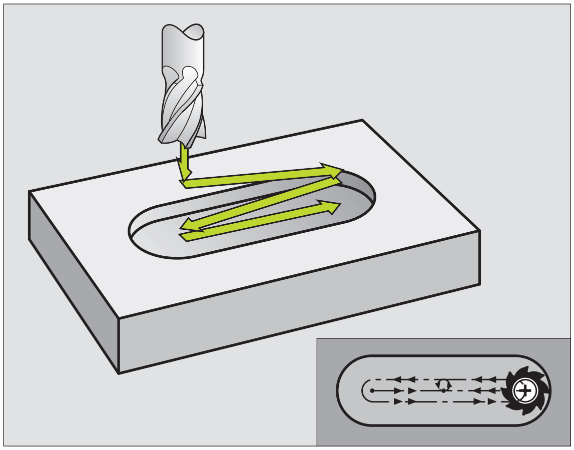
The TNC automatically pre-positions the tool in the tool axis and working plane.

During roughing the tool plunges into the material with a sideward reciprocating motion from one end of the slot to the other. Pilot drilling is therefore unnecessary.

The algebraic sign for the cycle parameter DEPTH determines the working direction. If you program DEPTH = 0, the cycle will not be executed.

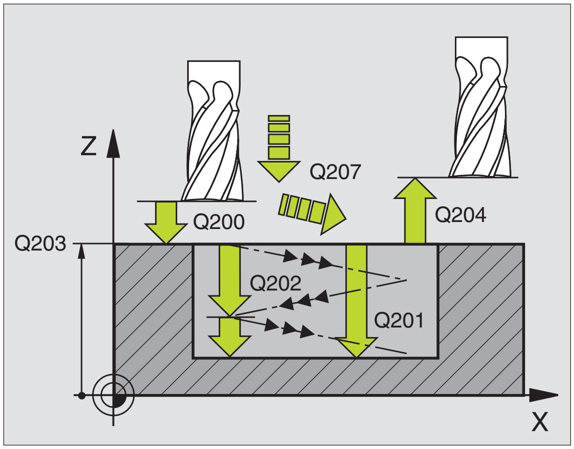
The cutter diameter must not be larger than the slot width and not smaller than a third of the slot width.

The cutter diameter must be smaller than half the slot length. The TNC otherwise cannot execute this cycle.



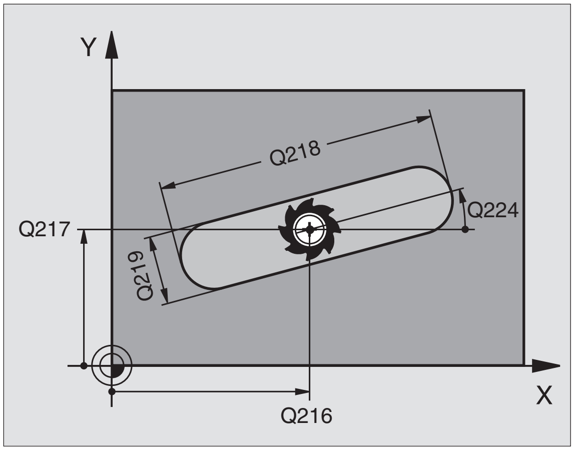
Roughing process

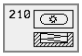
1 At rapid traverse, the TNC positions the tool in the tool axis to the 2nd set-up clearance and subsequently to the center of the left circle. From there, the TNC positions the tool to set-up clearance above the workpiece surface.
2 The tool moves at the feed rate for milling to the workpiece surface. From there, the cutter advances in the longitudinal direction of the slot—plunge-cutting obliquely into the material—until it reaches the center of the right circle.
3 The tool then moves back to the center of the left circle, again with oblique plunge-cutting. This process is repeated until the programmed milling depth is reached.
4 At the milling depth, the TNC moves the tool for the purpose of face milling to the other end of the slot and then back to the center of the slot.



Finishing process

5 The TNC advances the tool from the slot center tangentially to the contour of the finished part. The tool subsequently climb mills the contour (with M3), and if so entered, in more than one infeed.
6 When the tool reaches the end of the contour, it departs the contour tangentially and returns to the center of the slot.
7 At the end of the cycle, the tool is retracted at rapid traverse to the set-up clearance and—if programmed—to the 2nd set-up clearance.





- **`Set-up clearance`** Q200 (incremental value): Distance between tool tip and workpiece surface.
- **`Depth`** Q201 (incremental value): Distance between workpiece surface and bottom of slot.
- **`Feed rate for milling`** Q207: Traversing speed of the tool in mm/min while milling.
- **`Plunging depth`** Q202 (incremental value): Total extent by which the tool is fed in the tool axis during a reciprocating movement.
- **`Machining operation (0/1/2)`** Q215: Define the machining operation:
  0. Roughing and finishing
  1. Only roughing
  2. Only finishing
- **`Workpiece surface coordinate`** Q203 (absolute value): Coordinate of the workpiece surface.
- **`2nd set-up clearance`** Q204 (incremental value): Z coordinate at which no collision between tool and workpiece (clamping devices) can occur.
- **`Center in 1st axis`** Q216 (absolute value): Center of the slot in the reference axis of the working plane.
- **`Center in 2nd axis`** Q217 (absolute value): Center of the slot in the minor axis of the working plane.
- **`First side length`** Q218 (value parallel to the reference axis of the working plane): Enter the length of the slot
- **`Second side length`** Q219 (value parallel to the secondary axis of the working plane): Enter the slot width. If you enter a slot width that equals the tool diameter, the TNC will carry out the roughing process only (slot milling).
- **`Angle of rotation`** Q224 (absolute value): Angle by which the entire slot is rotated. The center of rotation lies in the center of the slot.

Not TNC 410

- **`Infeed for finishing`** Q338 (incremental value): Infeed per cut. Q338=0: Finishing in one infeed.

Example: NC block

```
N51 G210 Q200=2 Q201=-20 Q207=500 Q202=5
    Q215=0 Q203=+30 Q204=50 Q216=+50
    Q217=+50 Q218=80 Q219=12 Q224=+15
    Q338=5 *
```

CIRCULAR SLOT with reciprocating plunge-cut (Cycle G211)

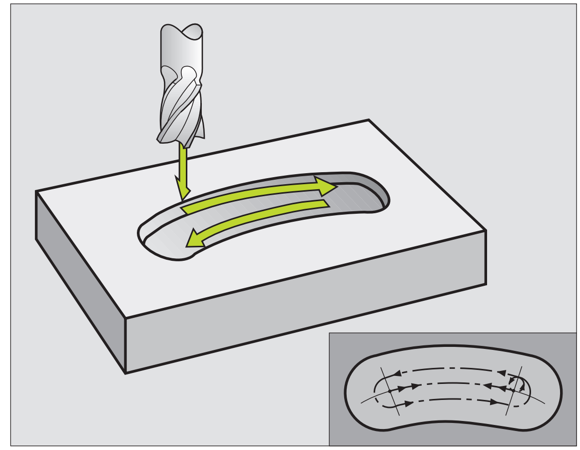
Roughing process

1 At rapid traverse, the TNC positions the tool in the tool axis to the 2nd set-up clearance and subsequently to the center of the right circle. From there, the tool is positioned to the programmed set-up clearance above the workpiece surface.

2 The tool moves at the milling feed rate to the workpiece surface. From there, the cutter advances—plunge-cutting obliquely into the material—to the other end of the slot.

3 The tool then moves at a downward angle back to the starting point, again with oblique plunge-cutting. This process (2 to 3) is repeated until the programmed milling depth is reached.

4 At the milling depth, the TNC moves the tool for the purpose of face milling to the other end of the slot.

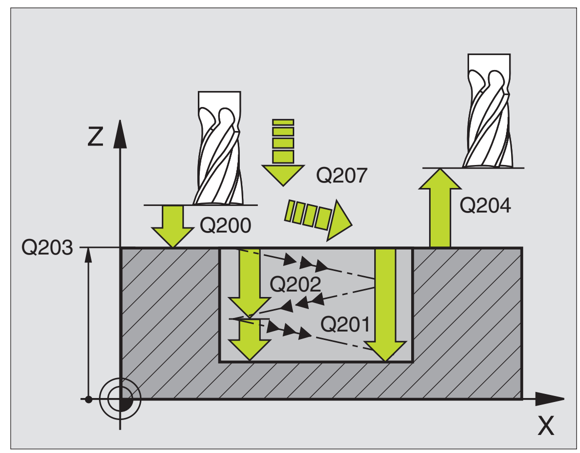


Finishing process

5 The TNC advances the tool from the slot center tangentially to the contour of the finished part. The tool subsequently climb mills the contour (with M3), and if so entered, in more than one infeed. The starting point for the finishing process is the center of the right circle.

6 When the tool reaches the end of the contour, it departs the contour tangentially.

7 At the end of the cycle, the tool is retracted at rapid traverse to the set-up clearance and—if programmed—to the 2nd set-up clearance.



Before programming, note the following:

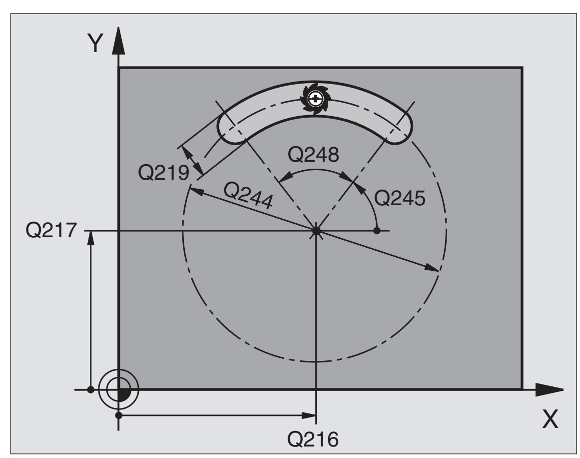
The TNC automatically pre-positions the tool in the tool axis and working plane.

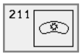
During roughing the tool plunges into the material with a helical sideward reciprocating motion from one end of the slot to the other. Pilot drilling is therefore unnecessary.

The algebraic sign for the cycle parameter DEPTH determines the working direction. If you program DEPTH = 0, the cycle will not be executed.

The cutter diameter must not be larger than the slot width and not smaller than a third of the slot width.

The cutter diameter must be smaller than half the slot length. The TNC otherwise cannot execute this cycle.





- **Set-up clearance** Q200 (incremental value): Distance between tool tip and workpiece surface.
- **Depth** Q201 (incremental value): Distance between workpiece surface and bottom of slot.
- **Feed rate for milling** Q207: Traversing speed of the tool in mm/min while milling.
- **Plunging depth** Q202 (incremental value): Total extent by which the tool is fed in the tool axis during a reciprocating movement.
- **Machining operation (0/1/2)** Q215: Define the machining operation:
  0. Roughing and finishing
  1. Only roughing
  2. Only finishing
- **Workpiece surface coordinate** Q203 (absolute value): Coordinate of the workpiece surface.
- **2nd set-up clearance** Q204 (incremental value): Z coordinate at which no collision between tool and workpiece (clamping devices) can occur.
- **Center in 1st axis** Q216 (absolute value): Center of the slot in the reference axis of the working plane.
- **Center in 2nd axis** Q217 (absolute value): Center of the slot in the minor axis of the working plane.
- **Pitch circle diameter** Q244: Enter the diameter of the pitch circle.
- **Second side length** Q219: Enter the slot width. If you enter a slot width that equals the tool diameter, the TNC will carry out the roughing process only (slot milling).
- **Starting angle** Q245 (absolute value): Enter the polar angle of the starting point.
- **Angular length** Q248 (incremental value): Enter the angular length of the slot.

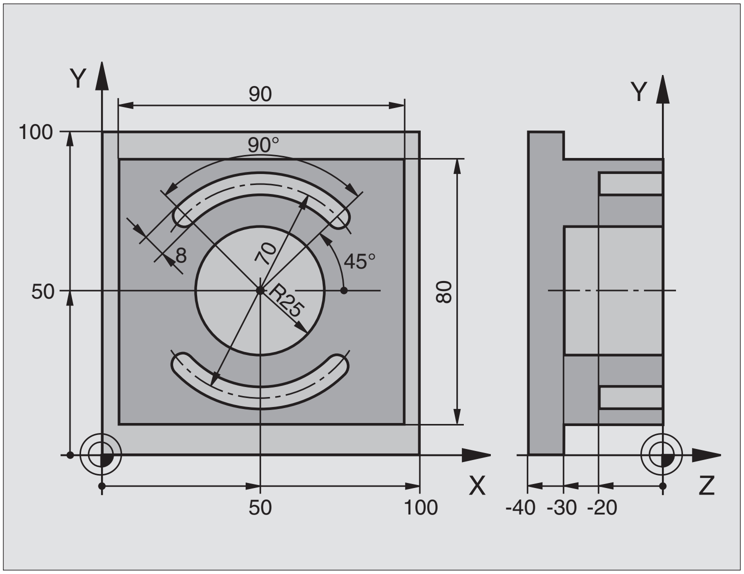
Not available with TNC 410:

- **Infeed for finishing** Q338 (incremental value): Infeed per cut. Q338=0: Finishing in one infeed.

Example: NC block

```
N52 G211 Q200=2 Q201=-20 Q207=500 Q202=5
    Q215=0 Q203=+30 Q204=50 Q216=+50
    Q217=+50 Q244=80 Q219=12 Q245=+45
    Q248=90 Q338=5 *
```

Example: Milling pockets, studs and slots



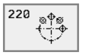
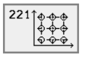
`%C210 G71 *`	
`N10 G30 G17 X+0 Y+0 Z-40 *`	Define the workpiece blank
`N20 G31 G90 X+100 Y+100 Z+0 *`	
`N30 G99 T1 L+0 R+6 *`	Define the tool for roughing/finishing
`N40 G99 T2 L+0 R+3 *`	Define slotting mill
`N50 T1 G17 S3500 *`	Call the tool for roughing/finishing
`N60 G00 G40 G90 Z+250 *`	Retract the tool
`N70 G213 Q200=2 Q201=-30 Q206=250 Q202=5`	Define cycle for machining the contour outside
`    Q207=250 Q203=+0 Q204=20 Q216=+50`	
`    Q217=+50 Q218+90 Q219=80 Q220=0 Q221=5*`	
`N80 G79 M03 *`	Call cycle for machining the contour outside
`N90 G78 P01 2 P02 -30 P03 5 P04 250 P05 25`	Define CIRCULAR POCKET MILLING cycle
`    P06 400 *`	
`N100 G00 G40 X+50 Y+50 *`	
`N110 Z+2 M99 *`	Call CIRCULAR POCKET MILLING cycle
`N120 Z+250 M06 *`	Tool change
`N130 T2 G17 S5000 *`	Call slotting mill

N140 G211 Q200=2 Q201=-20 Q207=250	Cycle definition for slot 1
Q202=5 Q215=0 Q203=+0 Q204=100	
Q216=+50 Q217=+50 Q244=70 Q219=8	
Q245=+45 Q248=90*	
N150 G79 M03 *	Call cycle for slot 1
N160 D00 Q245 P01 +225 *	New starting angle for slot 2
N170 G79 *	Call cycle for slot 2
N180 G00 Z+250 M02 *	Retract in the tool axis, end program
N999999 %C210 G71 *	

8.5 Cycles for Machining Hole Patterns

Overview

The TNC provides two cycles for machining hole patterns directly:

Cycle	Soft key
G220 CIRCULAR PATTERN	
G221 LINEAR PATTERN	

You can combine Cycle G220 and Cycle G221 with the following fixed cycles:

> If you have to machine irregular hole patterns, use **G79 "PAT"** to develop point tables (see “Point Tables” on page 180).

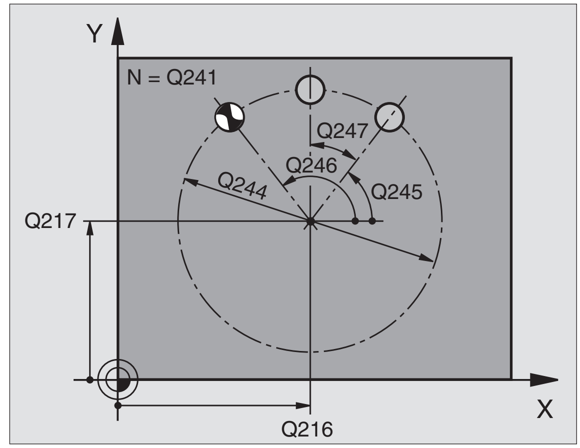
Cycle G83	PECKING
Cycle G84	TAPPING with a floating tap holder
Cycle G74	SLOT MILLING
Cycle G75/G76	POCKET MILLING
Cycle G77/G78	CIRCULAR POCKET MILLING
Cycle G85	RIGID TAPPING without a floating tap holder
Cycle G86	THREAD CUTTING

Cycle G200	DRILLING
Cycle G201	REAMING
Cycle G202	BORING
Cycle G203	UNIVERSAL DRILLING
Cycle G204	BACK BORING
Cycle G212	POCKET FINISHING
Cycle G213	STUD FINISHING
Cycle G214	CIRCULAR POCKET FINISHING
Cycle G215	CIRCULAR STUD FINISHING

Not available with TNC 410:

Cycle G205	UNIVERSAL PECKING
Cycle G206	TAPPING NEW with a floating tap holder
Cycle G207	RIGID TAPPING NEW without a floating tap holder
Cycle G208	BORE MILLING
Cycle G209	TAPPING WITH CHIP BREAKING
Cycle G262	THREAD MILLING
Cycle G263	THREAD MILLING/COUNTERSINKING
Cycle G264	THREAD DRILLING/MILLING
Cycle G265	HELICAL THREAD DRILLING/MILLING
Cycle G267	OUTSIDE THREAD MILLING

CIRCULAR PATTERN (Cycle G220)

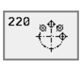


1. At rapid traverse, the TNC moves the tool from its current position to the starting point for the first machining operation.
   Sequence:
   - Move to second set-up clearance (spindle axis)
   - Approach the starting point in the spindle axis.
   - Move to set-up clearance above the workpiece surface (spindle axis).
2. From this position, the TNC executes the last defined fixed cycle.
3. The tool then approaches the starting point for the next machining operation on a straight line at set-up clearance (or 2nd set-up clearance).
4. This process (1 to 3) is repeated until all machining operations have been executed.

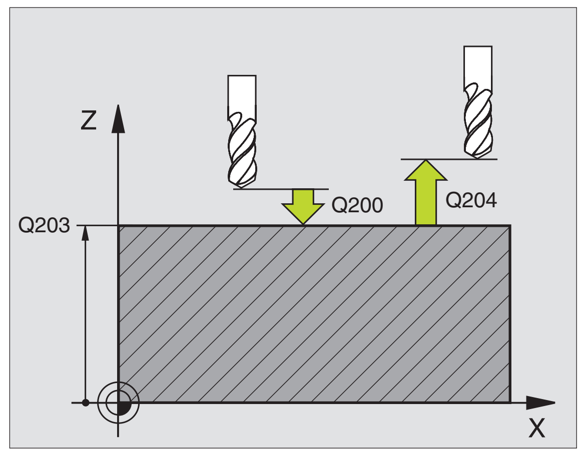
Before programming, note the following:

Cycle G220 is DEF active, which means that Cycle G220 automatically calls the last defined fixed cycle.

If you combine Cycle G220 with one of the fixed cycles G200 to G209, G212 to G215 and G262 to G267, the set-up clearance, workpiece surface and 2nd set-up clearance that you defined in Cycle G220 will be effective for the selected fixed cycle.



- **`Center in 1st axis`** Q216 (absolute value): Center of the pitch circle in the reference axis of the working plane.
- **`Center in 2nd axis`** Q217 (absolute value): Center of the pitch circle in the minor axis of the working plane.
- **`Pitch circle diameter`** Q244: Diameter of the pitch circle.
- **`Starting angle`** Q245 (absolute value): Angle between the reference axis of the working plane and the starting point for the first machining operation on the pitch circle.
- **`Stopping angle`** Q246 (absolute value): Angle between the reference axis of the working plane and the starting point for the last machining operation on the pitch circle (does not apply to complete circles). Do not enter the same value for the stopping angle and starting angle. If you enter the stopping angle greater than the starting angle, machining will be carried out counterclockwise; otherwise, machining will be clockwise.



Example: NC block

```
N53 G220 Q216=+50 Q217=+50 Q244=80
    Q245=+0 Q246=+360 Q247=+0 Q241=8
    Q200=2 Q203=+0 Q204=50 Q301=1 *
```

- **Stepping angle** Q247 (incremental value): Angle between two machining operations on a pitch circle. If you enter an angle step of 0, the TNC will calculate the angle step from the starting and stopping angles and the number of pattern repetitions. If you enter a value other than 0, the TNC will not take the stopping angle into account. The sign for the angle step determines the working direction (– = clockwise).
- **Number of repetitions** Q241: Number of machining operations on a pitch circle.
- **Set-up clearance** Q200 (incremental value): Distance between tool tip and workpiece surface. Enter a positive value.
- **Workpiece surface coordinate** Q203 (absolute value): Coordinate of the workpiece surface.
- **2nd set-up clearance** Q204 (incremental value): Coordinate in the tool axis at which no collision between tool and workpiece (clamping devices) can occur.

Not available with TNC 410:

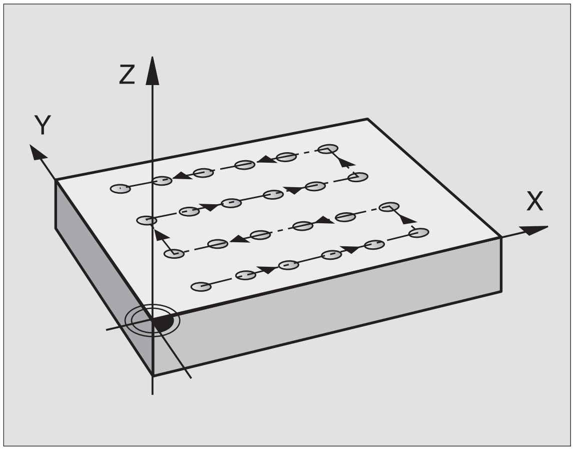
- **Traversing to clearance height** Q301: Definition of how the tool is to move between machining processes:
  0: Move to the set-up clearance between operations.
  1: Move to the 2nd set-up clearance between the measuring points.

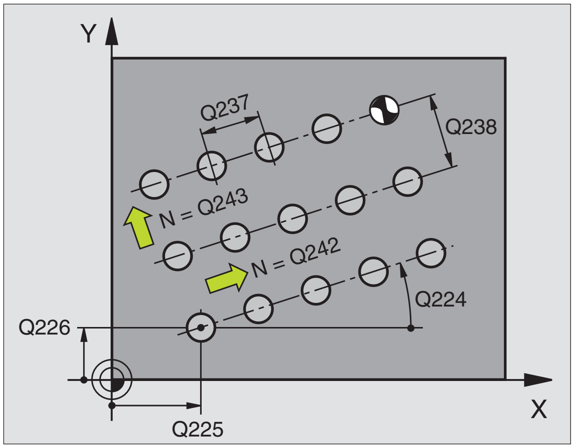
LINEAR PATTERN (Cycle G221)

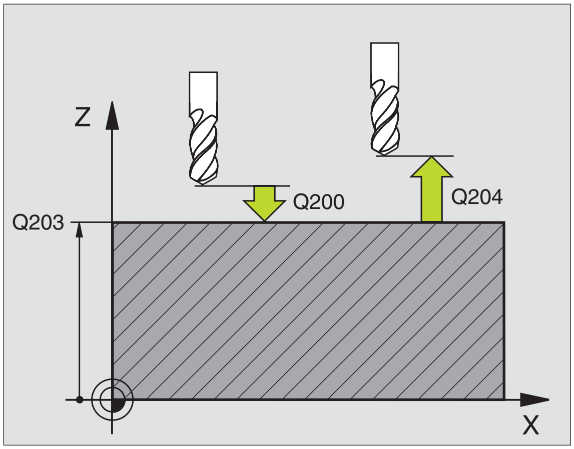
Before programming, note the following:

Cycle G221 is DEF active, which means that Cycle G221 automatically calls the last defined fixed cycle.

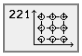
If you combine Cycle G221 with one of the fixed cycles G200 to G209, G212 to G215 and G262 to G267, the set-up clearance, workpiece surface and 2nd set-up clearance that you defined in Cycle G221 will be effective for the selected fixed cycle.







1 The TNC automatically moves the tool from its current position to the starting point for the first machining operation.
  Sequence:
  - Move to second set-up clearance (spindle axis)
  - Approach the starting point in the spindle axis.
  - Move to set-up clearance above the workpiece surface (spindle axis).
2 From this position, the TNC executes the last defined fixed cycle.
3 The tool then approaches the starting point for the next machining operation in the positive reference axis direction at set-up clearance (or 2nd set-up clearance).
4 This process (1 to 3) is repeated until all machining operations on the first line have been executed. The tool is located above the last point on the first line.
5 The tool subsequently moves to the last point on the second line where it carries out the machining operation.
6 From this position, the tool approaches the starting point for the next machining operation in the negative reference axis direction.
7 This process (6) is repeated until all machining operations in the second line have been executed.
8 The tool then moves to the starting point of the next line.
9 All subsequent lines are processed in a reciprocating movement.

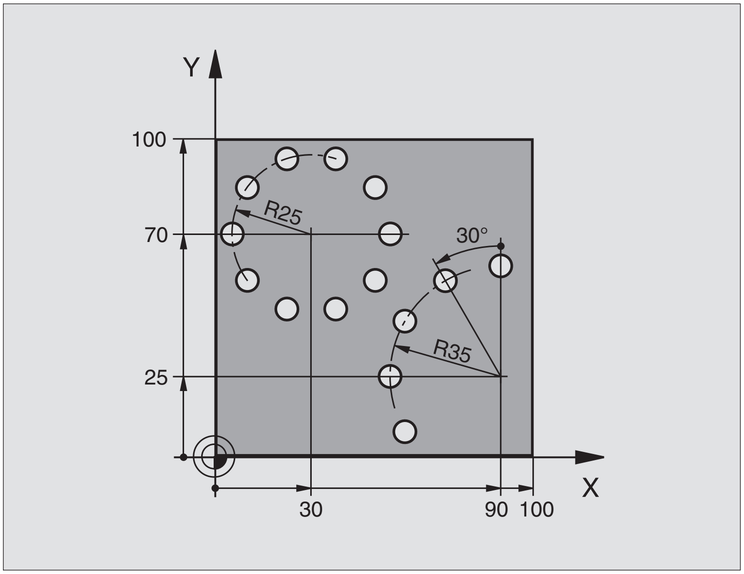


- **`Starting point 1st axis`** Q225 (absolute value): Coordinate of the starting point in the reference axis of the working plane.
- **`Starting point 2nd axis`** Q226 (absolute value): Coordinate of the starting point in the minor axis of the working plane.
- **`Spacing in 1st axis`** Q237 (incremental value): Spacing between the individual points on a line.
- **`Spacing in 2nd axis`** Q238 (incremental value): Spacing between the individual lines.
- **`Number of columns`** Q242: Number of machining operations on a line.
- **`Number of lines`** Q243: Number of passes.
- **`Angle of rotation`** Q224 (absolute value): Angle by which the entire pattern is rotated. The center of rotation lies in the starting point.
- **`Set-up clearance`** Q200 (incremental value): Distance between tool tip and workpiece surface.
- **`Workpiece surface coordinate`** Q203 (absolute value): Coordinate of the workpiece surface.
- **`2nd set-up clearance`** Q204 (incremental value): Coordinate in the tool axis at which no collision between tool and workpiece (clamping devices) can occur.
- **`Traversing to clearance height`** Q301: Definition of how the tool is to move between machining processes:
  0: Move to the set-up clearance between operations.
  1: Move to the 2nd set-up clearance between the measuring points.

Example: NC block

```
N54 G221 Q225=+15 Q226=+15 Q237=+10
    Q238=+8 Q242=6 Q243=4 Q224=+15
    Q200=2 Q203=+30 Q204=50 Q301=1 *
```

Example: Circular hole patterns



`%PATTERN G71 *`	
`N10 G30 G17 X+0 Y+0 Z-40 *`	Define the workpiece blank
`N20 G31 G90 X+100 Y+100 Z+0 *`	
`N30 G99 T1 L+0 R+3 *`	Define the tool
`N40 T1 G17 S3500 *`	Tool call
`N50 G00 G40 G90 Z+250 M03 *`	Retract the tool
`N60 G200 Q200=2 Q201=-15 Q206=250`	Cycle definition: drilling
`    Q202=4 Q210=0 Q203=+0 Q204=0 *`	
`N70 G220 Q216=+30 Q217=+70 Q244=50`	Cycle definition: circular hole pattern 1
`    Q245=+0 Q246=+360 Q247=+0 Q241=10`	
`    Q200=2 Q203=+0 Q204=100 *`	
`N80 G220 Q216=+90 Q217=+25 Q244=70`	Cycle definition: circular hole pattern 2
`    Q245=+90 Q246=+360 Q247=+30 Q241=5`	
`    Q200=2 Q203=+0 Q204=100 *`	
`N90 G00 G40 Z+250 M02 *`	Retract in the tool axis, end program
`N999999 %PATTERN G71`	

8.6 SL Cycles Group I

Fundamentals

SL Cycles enable you to form complex contours by combining up to 12 subcontours (pockets or islands). You define the individual subcontours in subprograms. The TNC calculates the total contour from the subcontours (subprogram numbers) that you enter in Cycle **G37** CONTOUR GEOMETRY.

> The memory capacity for programming an SL cycle (all contour subprograms) is limited to 48 kilobytes. The number of possible contour elements depends on the type of contour (inside or outside contour) and the number of subcontours. For example, you can program up to approx. 128 line blocks.

Characteristics of the subprograms

- Coordinate transformations are allowed. If they are programmed within the subcontour they are also effective in the following subprograms, but they need not be reset after the cycle call.
- The TNC ignores feed rates F and miscellaneous functions M.
- The TNC recognizes a pocket if the tool path lies inside the contour, for example if you machine the contour clockwise with radius compensation **G42.**
- The TNC recognizes an island if the tool path lies outside the contour, for example if you machine the contour clockwise with radius compensation **G41.**
- The subprograms must not contain tool axis coordinates.
- The working plane is defined in the first coordinate block of the subprogram. The secondary axes U,V,W are permitted.

Characteristics of the fixed cycles

> **TNC 410:**
>
> With MP7420.0 and MP7420.1 you can determine how the tool should be moved during rough-out (see "General User Parameters" on page 422).

- The TNC automatically positions the tool to the starting position in the machining plane before a cycle. In the spindle axis the tool must be pre-positioned to set-up clearance.
- Each level of infeed depth is roughed-out axis-parallel or at a preset angle (angle defined in Cycle **G57**). In the standard setting, islands are traversed at safety clearance. In MP7420.1 you can also define that the TNC should rough-out individual pockets separately, plunging only once for each pocket.

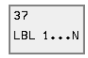
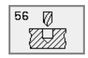
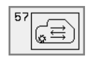
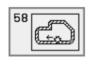
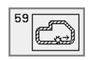
Example: Program structure: Machining with SL Cycles

```
%SL G71 *
...
N12 G37 P01 ...
...
N16 G56 P01 ...
N17 G79 *
...
N18 G57 P01 ...
N19 G79 *
...
N26 G59 P01 ...
N27 G79 *
...
N50 G00 G40 G90 Z+250 M2 *
N51 G98 L1 *
...
N60 G98 L0 *
N61 G98 L2 *
...
N62 G98 L0 *
...
N999999 %SL G71 *
```

- The TNC takes the entered finishing allowance (cycle **G57**) into consideration.

> With MP7420 you can determine where the tool is positioned at the end of Cycles 21 to 24.

Overview of SL Cycles, Group I

Cycle	Soft key
G37 CONTOUR GEOMETRY (essential)	
G56 PILOT DRILLING (optional)	
G57 ROUGH-OUT (essential)	
G58/G59 CONTOUR MILLING (optional) G58: In clockwise direction G59: In counterclockwise direction	 

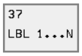
CONTOUR GEOMETRY (Cycle G37)

All subprograms that are superimposed to define the contour are listed in Cycle G37 CONTOUR GEOMETRY.

Before programming, note the following:

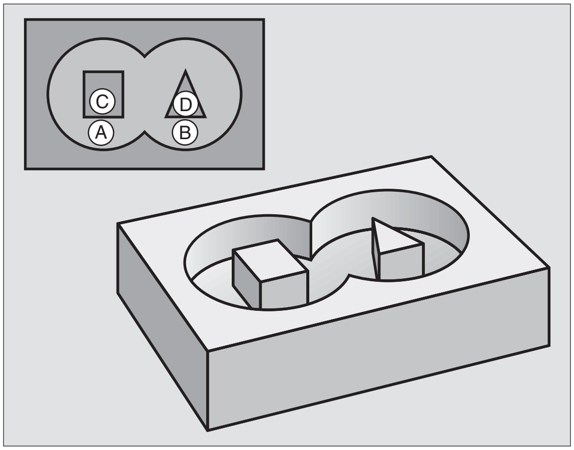
Cycle **`G37`** is DEF active which means that it becomes effective as soon as it is defined in the part program.

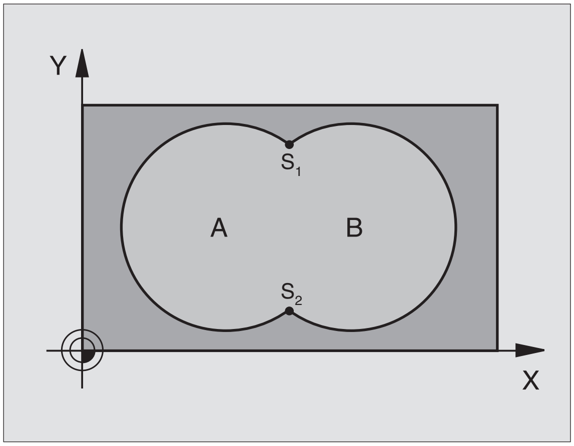
You can list up to 12 subprograms (subcontours) in Cycle **`G37`**.



- **`Label numbers for the contour:`** Enter all label numbers for the individual subprograms that are to be superimposed to define the contour. Confirm every label number with the ENT key. When you have entered all numbers, conclude entry with the END key.

Overlapping contours: (see "Overlapping contours" on page 267)





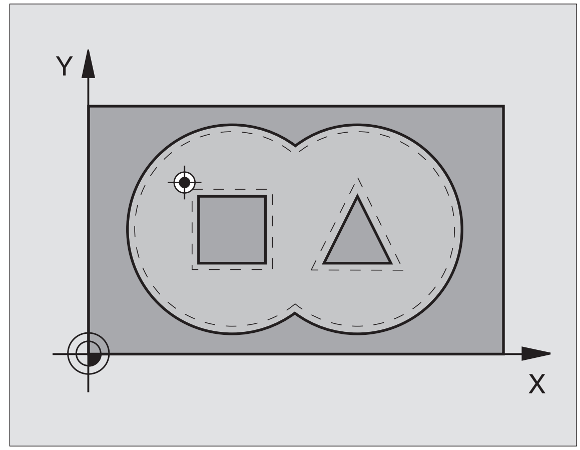
Example: NC blocks

```
N54 G37 P01 1  P02 5  P03 7  P04  8 *
```

PILOT DRILLING (Cycle G56)

Before programming, note the following:

Program a positioning block for the starting point in the tool axis (set-up clearance above the workpiece surface).

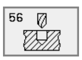


Process

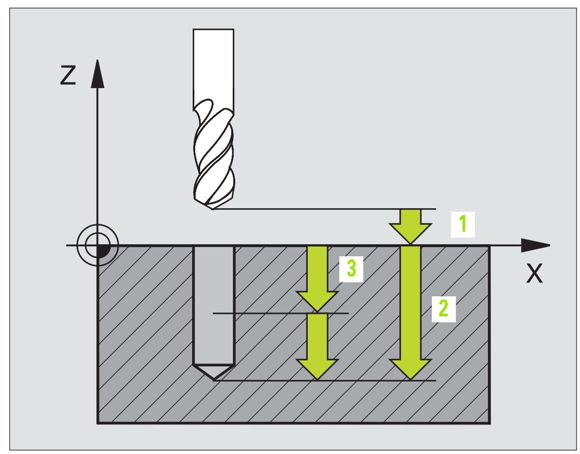
Same as Cycle **G83** Pecking; see “Cycles for Drilling, Tapping and Thread Milling,” page 183.

Application

Cycle **G56** is for PILOT DRILLING of the cutter infeed points. It accounts for the finishing allowance. The cutter infeed points also serve as starting points for roughing.



- **Set-up clearance** 1 (incremental value): Distance between tool tip (at starting position) and workpiece surface
- **Total hole depth** 2 (incremental value): Distance between workpiece surface and bottom of hole (tip of drill taper)
- **Plunging depth** 3 (incremental value): Infeed per cut The total hole depth does not have to be a multiple of the plunging depth. The tool will drill to the total hole depth in one movement if:
  - the plunging depth is equal to the depth
  - the plunging depth is greater than the total hole depth
- **Feed rate for plunging**: Traversing speed in mm/min for drilling
- **Finishing allowance:** Allowance in the machining plane



Example: NC blocks

```
N54 G56 P01 2  P02 -15  P03 5  P04 250
    P05 +0.5*
```

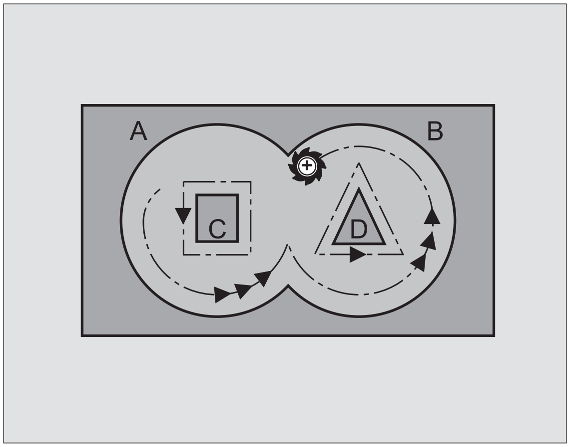
ROUGH-OUT (Cycle G57)

Process

1 The TNC positions the tool in the working plane above the first cutting point, taking the finishing allowance into consideration.
2 The TNC moves the tool at the feed rate for plunging to the first plunging depth.

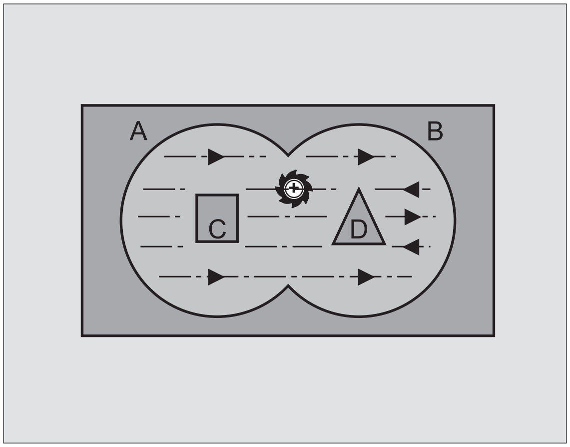
The contour is fully rough-milled (see figure at top right):

1 The tool mills the first subcontour at the programmed feed rate, taking the finishing allowance in the machining plane into consideration.
2 Further depths and further subcontours are milled by the TNC in the same way.
3 The TNC moves the tool in the spindle axis to the set-up clearance and then positions it above the first cutter infeed point in the machining plane.



Rough out pocket (see figure at center right):

1 After reaching the first plunging depth, the tool mills the contour at the programmed feed rate paraxially or at the entered roughing angle.
2 The island contours (here: C/D) are traversed at set-up clearance.
3 This process is repeated until the programmed milling depth is reached.

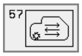


Before programming, note the following:

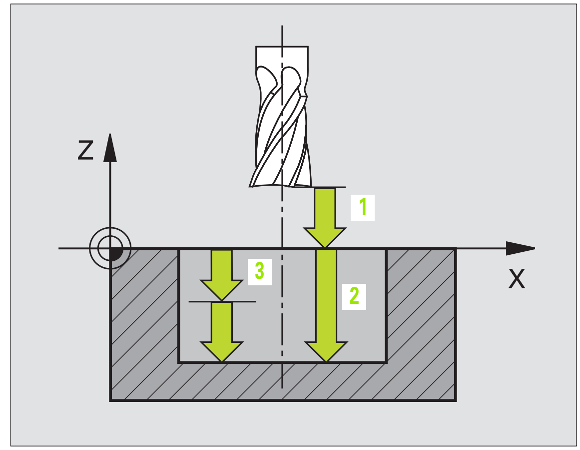
With MP7420.0 and MP7420.1 you define how the TNC should machine the contour (see "General User Parameters" on page 422).

Program a positioning block for the starting point in the tool axis (set-up clearance above the workpiece surface).

This cycle requires a center-cut end mill (ISO 1641) or pilot drilling with Cycle 21.



- **Set-up clearance 1** (incremental value): Distance between tool tip (at starting position) and workpiece surface
- **Milling depth 2** (incremental value): Distance between workpiece surface and bottom of pocket
- **Plunging depth 3** (incremental value): Infeed per cut The milling depth does not have to be a multiple of the plunging depth. The TNC will go to depth in one movement if:
  - the plunging depth is equal to the depth
  - The plunging depth is greater than the milling depth
- **Feed rate for plunging:** Traversing speed of the tool in mm/min during penetration
- **Finishing allowance:** Allowance in the machining plane
- **Rough-out angle**: Direction of the roughing-out movement The rough-out angle is referenced to the reference axis of the machining plane. Enter the angle so that the cuts can be as long as possible.
- **Feed rate:** Feed rate for milling in mm/min



Example: NC block

```
N54 G57 P01 2  P02 -15  P03 5  P04 250
    P05 +0.5  P06 +30  P07 500 *
```

CONTOUR MILLING (Cycle G58/G59)

Before programming, note the following:

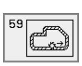
Program a positioning block for the starting point in the tool axis (set-up clearance above the workpiece surface).

Application

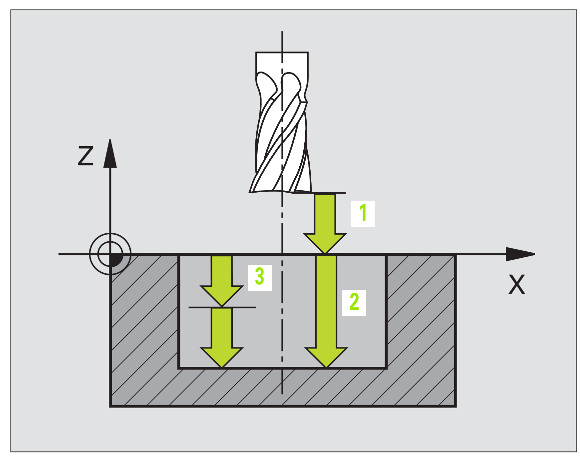
Cycle G58/G59 CONTOUR MILLING serves for finishing the contour pocket.

Direction of rotation during contour milling

- In clockwise direction: **G58**
- In counterclockwise direction: **G59**



- **Set-up clearance 1** (incremental value): Distance between tool tip (at starting position) and workpiece surface
- **Milling depth 2** (incremental value): Distance between workpiece surface and bottom of pocket
- **Plunging depth 3** (incremental value): Infeed per cut The milling depth does not have to be a multiple of the plunging depth. The TNC will go to depth in one movement if:
  - the plunging depth is equal to the depth
  - The plunging depth is greater than the milling depth
- **Feed rate for plunging:** Traversing speed of the tool in mm/min during penetration
- **Feed rate:** Feed rate for milling in mm/min



Example: NC blocks

```
N54 G58 P01 2  P02 -15  P03 5  P04 250
    P05 500*
...
N71 G59 P01 2  P02 -15 P03  5 P04  250
    P05 500*
```

8.7 SL Cycles Group II (not TNC 410)

Fundamentals

SL Cycles enable you to form complex contours by combining up to 12 subcontours (pockets or islands). You define the individual subcontours in subprograms. The TNC calculates the total contour from the subcontours (subprogram numbers) that you enter in Cycle **G37** CONTOUR GEOMETRY.

> The memory capacity for programming an SL cycle (all contour subprograms) is limited to 48 kilobytes. The number of possible contour elements depends on the type of contour (inside or outside contour) and the number of subcontours. For example, you can program up to approx. 256 line blocks.

Characteristics of the subprograms

- Coordinate transformations are allowed. If they are programmed within the subcontour they are also effective in the following subprograms, but they need not be reset after the cycle call.
- The TNC ignores feed rates F and miscellaneous functions M.
- The TNC recognizes a pocket if the tool path lies inside the contour, for example if you machine the contour clockwise with radius compensation **G42.**
- The TNC recognizes an island if the tool path lies outside the contour, for example if you machine the contour clockwise with radius compensation **G41.**
- The subprograms must not contain tool axis coordinates.
- The working plane is defined in the first coordinate block of the subprogram. The secondary axes U,V,W are permitted.

Characteristics of the fixed cycles

- The TNC automatically positions the tool to set-up clearance before a cycle.
- Each level of infeed depth is milled without interruptions since the cutter traverses around islands instead of over them.
- The radius of "inside corners" can be programmed—the tool keeps moving to prevent surface blemishes at inside corners (this applies for the outermost pass in the Rough-out and Side-Finishing cycles).
- The contour is approached in a tangential arc for side finishing.
- For floor finishing, the tool again approaches the workpiece in a tangential arc (for tool axis Z, for example, the arc may be in the Z/X plane).
- The contour is machined throughout in either climb or up-cut milling.

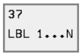
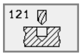
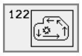
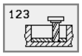
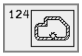
> With MP7420 you can determine where the tool is positioned at the end of Cycles G121 to G124.

The machining data (such as milling depth, finishing allowance and set-up clearance) are entered as CONTOUR DATA in Cycle **G120.**

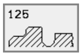
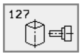
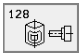
Example: Program structure: Machining with SL Cycles

```
%SL2 G71 *
...
N120 G37 ... *
N130 G120... *
...
N160 G121 ... *
N170 G79 *
...
N180 G122 ... *
N190 G79 *
...
N220 G123 ... *
N230 G79 *
...
N260 G124 ... *
N270 G79 *
...
N500 G00 G40 Z+250 M2 *
N510 G98 L1 *
...
N550 G98 L0 *
N560 G98 L2 *
...
N600 G98 L0 *
...
N99999 %SL2 G71 *
```

Overview of SL Cycles

Cycle	Soft key
G37 CONTOUR GEOMETRY (essential)	
G120 CONTOUR DATA (essential)	
G121 PILOT DRILLING (optional)	
G122 ROUGH-OUT (essential)	
G123 FLOOR FINISHING (optional)	
G124 SIDE FINISHING (optional)	

Enhanced cycles:

Cycle	Soft key
G125 CONTOUR TRAIN	
G127 CYLINDER SURFACE	
G128 CYLINDER SURFACE slot milling	

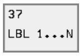
CONTOUR GEOMETRY (Cycle G37)

All subprograms that are superimposed to define the contour are listed in Cycle **G37** CONTOUR GEOMETRY.

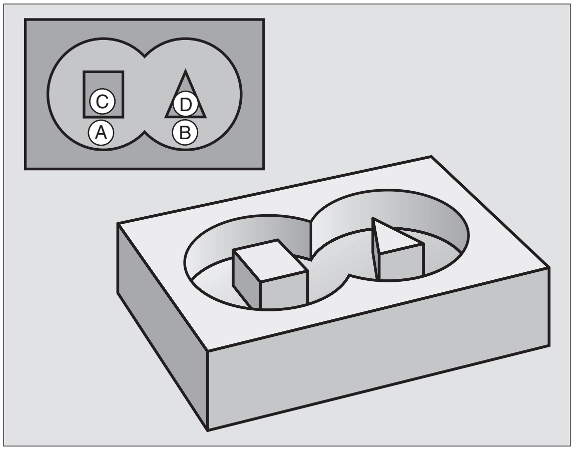
Before programming, note the following:

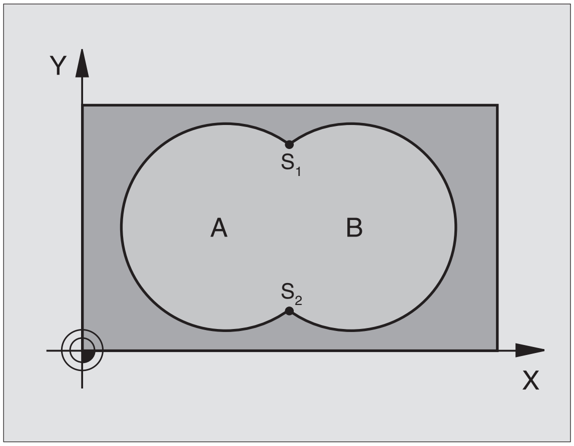
Cycle **G37** is DEF active which means that it becomes effective as soon as it is defined in the part program.

You can list up to 12 subprograms (subcontours) in Cycle **G37**.



- **Label numbers for the contour:** Enter all label numbers for the individual subprograms that are to be superimposed to define the contour. Confirm every label number with the ENT key. When you have entered all numbers, conclude entry with the END key.





Example: NC blocks

```
N120 G37 P01 1  P02 5  P03 7  P04 8 *
```

Overlapping contours

Pockets and islands can be overlapped to form a new contour. You can thus enlarge the area of a pocket by another pocket or reduce it by an island.

Subprograms: Overlapping pockets

The subsequent programming examples are contour subprograms that are called by Cycle **G37** CONTOUR GEOMETRY in a main program.

Pockets A and B overlap.

The TNC calculates the points of intersection S1 and S2 (they do not have to be programmed).

The pockets are programmed as full circles.

Subprogram 1: Pocket A

```
N510 G98 L1 *
N520 G01 G42 X+10 Y+50 *
N530 I+35 J+50 *
N540 G02 X+10 Y+50  *
N550 G98 L0 *
```

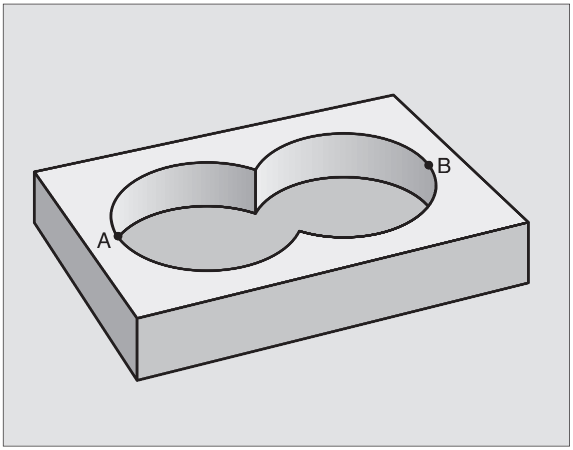
Subprogram 2: Pocket B

```
N560 G98 L2 *
N570 G01 G42 X+90 Y+50 *
N580 I+65 J+50  *
N590 G02 X+90 Y+50 *
N600 G98 L0 *
```

Area of inclusion

Both surfaces A and B are to be machined, including the mutually overlapped area:

- The surfaces A and B must be pockets.
- The first pocket (in Cycle **G37**) must start outside the second pocket.



Surface A:

```
N510 G98 L1 *
N520 G01 G42 X+10 Y+50 *
N530 I+35 J+50 *
N540 G02 X+10 Y+50  *
N550 G98 L0 *
```

Surface B:

```
N560 G98 L2 *
N570 G01 G42 X+90 Y+50 *
N580 I+65 J+50  *
N590 G02 X+90 Y+50 *
N600 G98 L0 *
```

Area of exclusion

Surface A is to be machined without the portion overlapped by B:

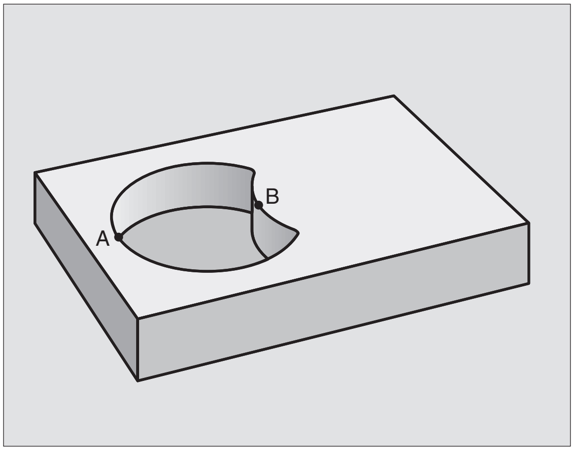
- Surface A must be a pocket and B an island.
- A must start outside of B.

Surface A:

```
N510 G98 L1 *
N520 G01 G42 X+10 Y+50 *
N530 I+35 J+50 *
N540 G02 X+10 Y+50  *
N550 G98 L0 *
```

Surface B:

```
N560 G98 L2 *
N570 G01 G41 X+90 Y+50 *
N580 I+65 J+50  *
N590 G02 X+90 Y+50 *
N600 G98 L0 *
```



Area of intersection

Only the area overlapped by both A and B is to be machined. (The areas covered by A or B alone are to be left unmachined.)

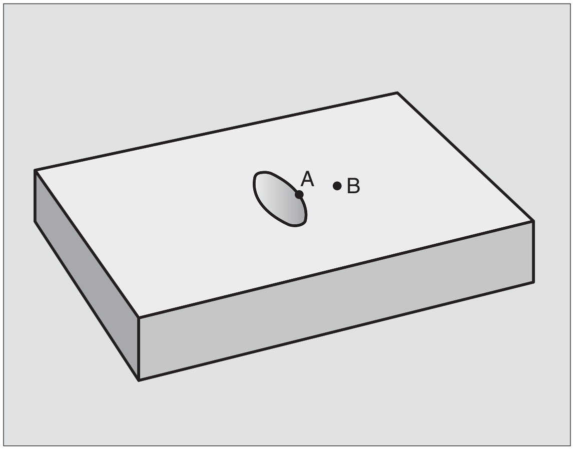
- A and B must be pockets.
- A must start inside of B.

Surface A:

```
N510 G98 L1 *
N520 G01 G42 X+60 Y+50 *
N530 I+35 J+50 *
N540 G02 X+60 Y+50 *
N550 G98 L0 *
```

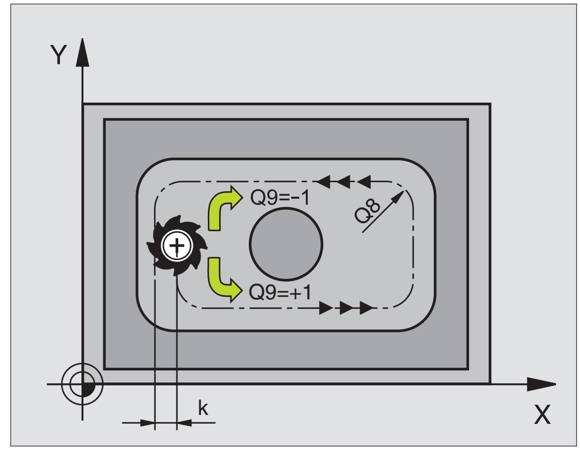
Surface B:

```
N560 G98 L2 *
N570 G01 G42 X+90 Y+50 *
N580 I+65 J+50  *
N590 G02 X+90 Y+50 *
N600 G98 L0 *
```

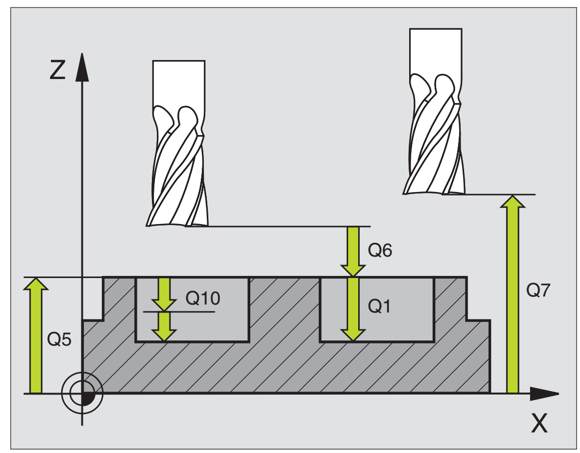


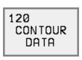
CONTOUR DATA (Cycle G120)

Machining data for the subprograms describing the subcontours are entered in Cycle **G120.**



> **Before programming, note the following:**
>
> Cycle **G120** is DEF active, meaning Cycle **G120**becomes effective as soon as it is defined in the part program.
>
> The algebraic sign for the cycle parameter DEPTH determines the working direction. If you program depth = 0, the TNC does not execute that next cycle.
>
> The machining data entered in Cycle **G120** are valid for Cycles G121 to G124.
>
> If you are using the SL cycles in Q parameter programs, the cycle parameters Q1 to Q19 cannot be used as program parameters.





- **Milling depth** Q1 (incremental value): Distance between workpiece surface and bottom of pocket.
- **Path overlap** factor Q2: Q2 x tool radius = stepover factor k.
- **Finishing allowance for side** Q3 (incremental value): Finishing allowance in the working plane
- **Finishing allowance for floor** Q4 (incremental value): Finishing allowance in the tool axis.
- **Workpiece surface coordinate** Q5 (absolute value): Absolute coordinate of the workpiece surface
- **Set-up clearance** Q6 (incremental value): Distance between tool tip and workpiece surface.
- **Clearance height** Q7 (absolute value): Absolute height at which the tool cannot collide with the workpiece (for intermediate positioning and retraction at the end of the cycle).
- **Inside corner radius** Q8: Inside "corner" rounding radius; entered value is referenced to the tool midpoint path.
- **Direction of rotation ? Clockwise = -1** Q9: Machining direction for pockets.
  - Clockwise (Q9 = –1 up-cut milling for pocket and island)
  - Counterclockwise (Q9 = +1 climb milling for pocket and island)

You can check the machining parameters during a program interruption and overwrite them if required.

Example: NC block

```
N57 G120 Q1=-20 Q2=1 Q3=+0.2 Q4=+0.1 Q5=+30
    Q6=+2 Q7=+80 Q8=0.5 Q9=+1 *
```

PILOT DRILLING (Cycle G121)

When calculating the infeed points, the TNC does not account for the delta value **DR** programmed in a **T** block.

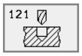
In narrow areas, the TNC may not be able to carry out pilot drilling with a tool that is larger than the rough-out tool.

Process

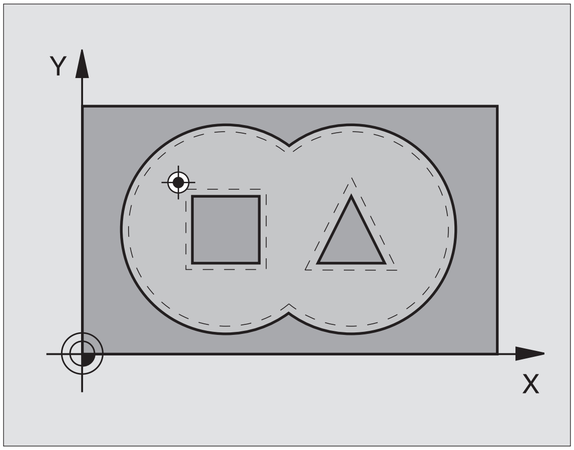
Same as Cycle **G83** Pecking; see "Cycles for Drilling, Tapping and Thread Milling," page 183.

Application

Cycle **G121** is for PILOT DRILLING of the cutter infeed points. It accounts for the allowance for side and the allowance for floor as well as the radius of the rough-out tool. The cutter infeed points also serve as starting points for roughing.



- **Plunging depth** Q10 (incremental value): Dimension by which the tool drills in each infeed (negative sign for negative working direction).
- **Feed rate for plunging** Q11: Traversing speed in mm/min during drilling.
- **Rough-out tool number** Q13: Tool number of the roughing mill.

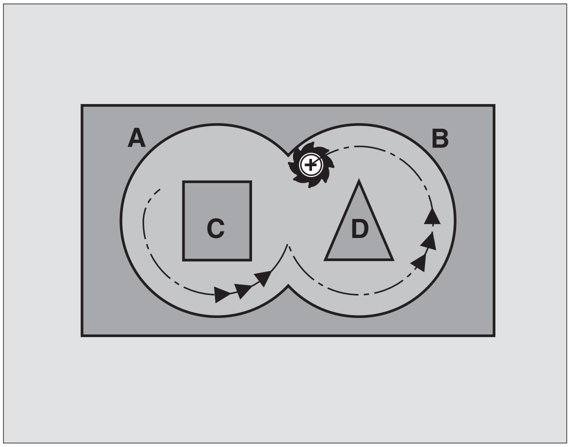


Example: NC blocks

```
N58 G121 Q10=+5 Q11=100 Q13=1 *
```

ROUGH-OUT (Cycle G122)

1. The TNC positions the tool over the cutter infeed point, taking the allowance for side into account.
2. In the first plunging depth, the tool mills the contour from inside outward at the milling feed rate Q12.
3. The island contours (here: C/D) are cleared out with an approach toward the pocket contour (here: A/B).
4. Then the TNC rough-mills the pocket contour retracts the tool to the clearance height.



Before programming, note the following:

This cycle requires a center-cut end mill (ISO 1641) or pilot drilling with Cycle **`G121`**.

- **`Plunging depth`** Q10 (incremental value): Dimension by which the tool plunges in each infeed.
- **`Feed rate for plunging`** Q11: Traversing speed of the tool in mm/min during penetration.
- **`Feed rate for milling`** Q12: Traversing speed for milling in mm/min.
- **`Coarse roughing tool number`** Q18: Number of the tool with which the TNC has already coarse-roughed the contour. If there was no coarse roughing, enter "0"; if you enter a value other than zero, the TNC will only rough-out the portion that could not be machined with the coarse roughing tool.
  If the portion that is to be roughed cannot be approached from the side, the TNC will mill in a reciprocating plunge-cut; For this purpose you must enter the tool length LCUTS in the tool table TOOL.T (see "Tool Data," page 99) and define the maximum plunging ANGLE of the tool. The TNC will otherwise generate an error message.
- **`Reciprocation feed rate`** Q19: Traversing speed of the tool in mm/min during reciprocating plunge-cut.

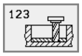
Example: NC block

```
N57 G120 Q10=+5 Q11=100 Q12=350 Q18=1
    Q19=150 *
```

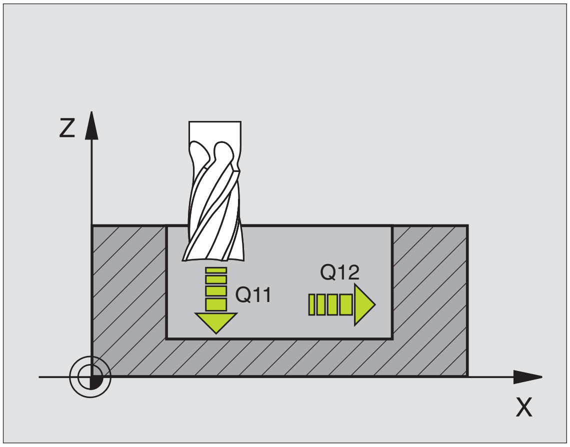
FLOOR FINISHING (Cycle G123)

> The TNC automatically calculates the starting point for finishing. The starting point depends on the available space in the pocket.

The tool approaches the machining plane smoothly (in a vertically tangential arc). The tool then clears the finishing allowance remaining from rough-out.



- **Feed rate for plunging** Q11: Traversing speed of the tool during penetration.
- **Feed rate for milling** Q12: Traversing speed for milling.



Example: NC block

```
N60 G123 Q11=100 Q12=350 *
```

SIDE FINISHING (Cycle G124)

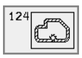
The subcontours are approached and departed on a tangential arc. Each subcontour is finish-milled separately.

Before programming, note the following:

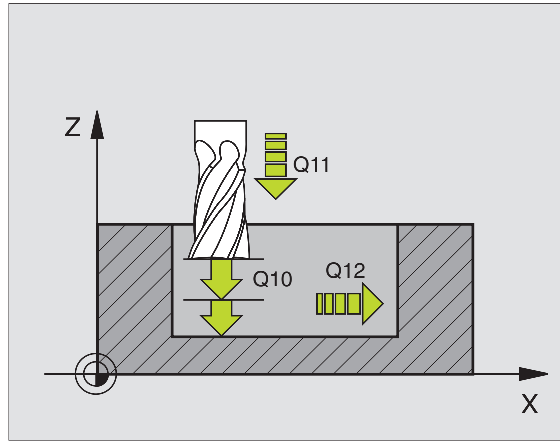
The sum of allowance for side (Q14) and the radius of the finish mill must be smaller than the sum of allowance for side (Q3, Cycle **G120**) and the radius of the rough mill.

This calculation also holds if you run Cycle **G124** without having roughed out with Cycle **G122**; in this case, enter "0" for the radius of the rough mill.

The TNC automatically calculates the starting point for finishing. The starting point depends on the available space in the pocket.



- **Direction of rotation ? Clockwise = -1** Q9: Machining direction:
  +1: Counterclockwise
  -1: Clockwise
- **Plunging depth** Q10 (incremental value): Dimension by which the tool plunges in each infeed.
- **Feed rate for plunging** Q11: Traversing speed of the tool during penetration.
- **Feed rate for milling** Q12: Traversing speed for milling.
- **Finishing allowance for side** Q14 (incremental value): Enter the allowed material for several finish-milling operations. If you enter Q14 = 0, the remaining finishing allowance will be cleared.



Example: NC block

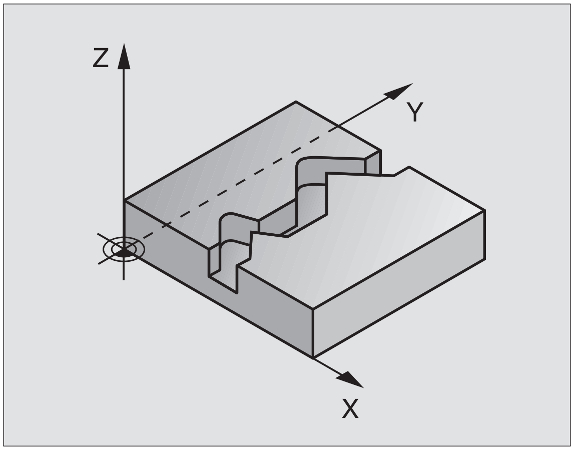
```
N61 G124 Q9=+1 Q10=+5 Q11=100 Q12=350
    Q14=+0 *
```

CONTOUR TRAIN (Cycle G125)

In conjunction with Cycle **G37** CONTOUR GEOMETRY, this cycle facilitates the machining of open contours (i.e. where the starting point of the contour is not the same as its end point).

Cycle **G125** CONTOUR TRAIN offers considerable advantages over machining an open contour using positioning blocks:

- The TNC monitors the operation to prevent undercuts and surface blemishes. It is recommended that you run a graphic simulation of the contour before execution.
- If the radius of the selected tool is too large, the corners of the contour may have to be reworked.
- The contour can be machined throughout by up-cut or by climb milling. The type of milling even remains effective when the contours are mirrored.
- The tool can traverse back and forth for milling in several infeeds: This results in faster machining.
- Allowance values can be entered in order to perform repeated rough-milling and finish-milling operations.



Before programming, note the following:

The algebraic sign for the cycle parameter DEPTH determines the working direction. If you program DEPTH = 0, the cycle will not be executed.

The TNC takes only the first label of Cycle **G37** CONTOUR GEOMETRY into account.

The memory capacity for programming an SL cycle is limited. For example, you can program up to 128 straight-line blocks in one SL cycle.

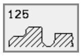
Cycle **G120** CONTOUR DATA is not required.

Positions that are programmed in incremental dimensions immediately after Cycle **G125** are referenced to the position of the tool at the end of the cycle.

Danger of collision!

To avoid collisions,

- Do not program positions in incremental dimensions immediately after Cycle **G125,** since they are referenced to the position of the tool at the end of the cycle.
- Move the tool to defined (absolute) positions in all main axes, since the position of the tool at the end of the cycle is not identical to the position of the tool at the start of the cycle.



- **`Milling depth`** Q1 (incremental value): Distance between workpiece surface and contour floor.
- **`Finishing allowance for side`** Q3 (incremental value): Finishing allowance in the working plane.
- **`Workpiece surface coordinate`** Q5 (absolute value): Absolute coordinate of the workpiece surface referenced to the workpiece datum.
- **`Clearance height`** Q7 (absolute value): Absolute height at which the tool cannot collide with the workpiece. Position for tool retraction at the end of the cycle.
- **`Plunging depth`** Q10 (incremental value): Dimension by which the tool plunges in each infeed.
- **`Feed rate for plunging`** Q11: Traversing speed of the tool in the tool axis.
- **`Feed rate for milling`** Q12: Traversing speed of the tool in the working plane.
- **`Climb or up-cut ?  Up-cut = –1`** Q15:
  Climb milling: Input value = +1
  Up-cut milling: Input value = –1
  To enable climb milling and up-cut milling alternately in several infeeds:Input value = 0

Example: NC block

```
N62 G125 Q1=-20 Q3=+0 Q5=+0 Q7=+50 Q10=+5
    Q11=100 Q12=350 Q15=+1 *
```

CYLINDER SURFACE (Cycle G127)

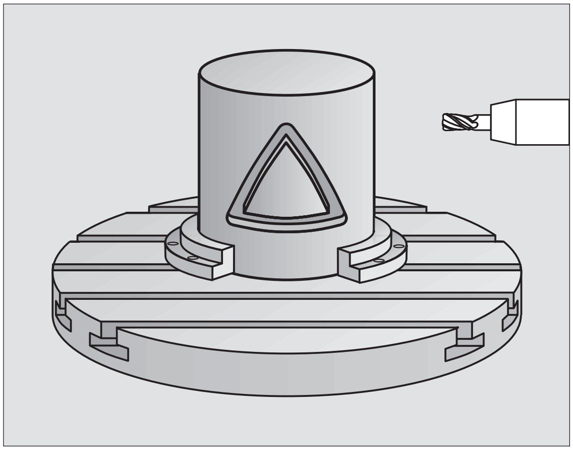
Machine and control must be specially prepared by the machine tool builder for use of this cycle.

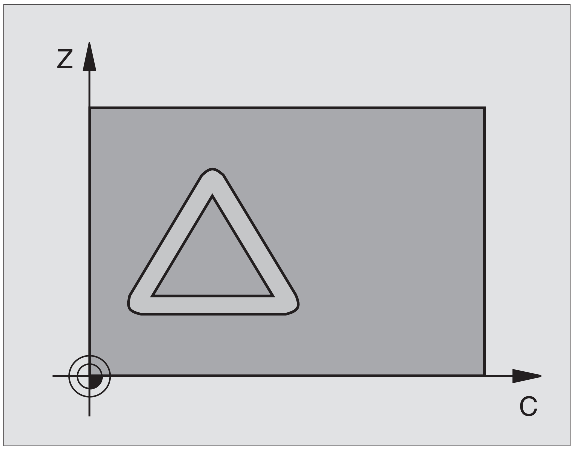
This cycle enables you to program a contour in two dimensions and then roll it onto a cylindrical surface for 3-D machining. Use Cycle **G128** if you wish to mill guide notches onto the cylinder surface.

The contour is described in a subprogram identified in Cycle **G37** CONTOUR GEOMETRY.

The subprogram contains coordinates in a rotary axis and in its parallel axis. The rotary axis C, for example, is parallel to the Z axis. The available path functions are G1, G11, G24, G25 and G2/G3/G12/G13 with R.

The dimensions in the rotary axis can be entered as desired either in degrees or in mm (or inches). You can select the desired dimension type in the cycle definition.





1 The TNC positions the tool over the cutter infeed point, taking the allowance for side into account.

2 At the first plunging depth, the tool mills along the programmed contour at the milling feed rate Q12.

3 At the end of the contour, the TNC returns the tool to the setup clearance and returns to the point of penetration.

4 Steps 1 to 3 are repeated until the programmed milling depth Q1 is reached.

5 Then the tool moves to the setup clearance.

Before programming, note the following:

The memory capacity for programming an SL cycle is limited. For example, you can program up to 256 straight-line blocks in one SL cycle.

The algebraic sign for the cycle parameter DEPTH determines the working direction. If you program DEPTH = 0, the cycle will not be executed.

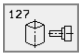
This cycle requires a center-cut end mill (ISO 1641).

The cylinder must be set up centered on the rotary table.

The tool axis must be perpendicular to the rotary table. If this is not the case, the TNC will generate an error message.

This cycle can also be used in a tilted working plane.

The TNC checks whether the compensated and non-compensated tool paths lie within the display range of the rotary axis, which is defined in Machine Parameter 810.x. If the error message “Contour programming error” is output, set MP 810.x = 0.



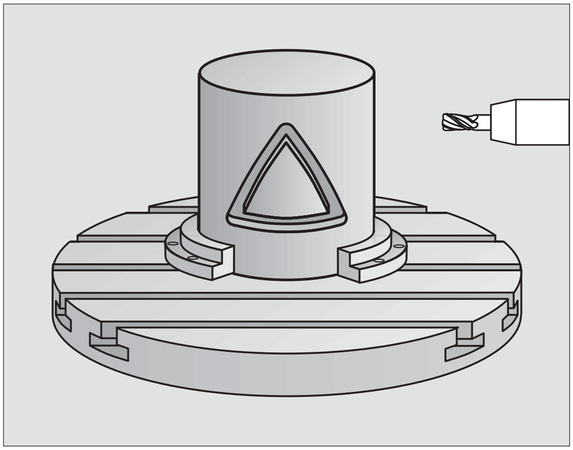
- **`Milling depth`** Q1 (incremental value): Distance between the cylindrical surface and the floor of the contour.
- **`Finishing allowance for side`** Q3 (incremental value): Finishing allowance in the plane of the unrolled cylindrical surface. This allowance is effective in the direction of the radius compensation.
- **`Set-up clearance`** Q6 (incremental value): Distance between the tool tip and the cylinder surface.
- **`Plunging depth`** Q10 (incremental value): Dimension by which the tool plunges in each infeed.
- **`Feed rate for plunging`** Q11: Traversing speed of the tool in the tool axis.
- **`Feed rate for milling`** Q12: Traversing speed of the tool in the working plane.
- **`Cylinder radius`** Q16: Radius of the cylinder on which the contour is to be machined.
- **`Dimension type ? ang./lin.`** Q17: The dimensions for the rotary axis of the subprogram are given either in degrees (0) or in mm/inches (1).

Example: NC block

```
N63 G127 Q1=-8 Q3=+0 Q6=+0 Q10=+3 Q11=100
    Q12=350 Q16=25 Q17=0 *
```

CYLINDER SURFACE slot milling (Cycle G128)

Machine and control must be specially prepared by the machine tool builder for use of this cycle.



This cycle enables you to program a guide notch in two dimensions and then transfer it onto a cylindrical surface. Unlike Cycle **G127**, with this cycle the TNC adjusts the tool so that, with radius compensation active, the walls of the slot are always parallel. Program the center-line path of the contour.

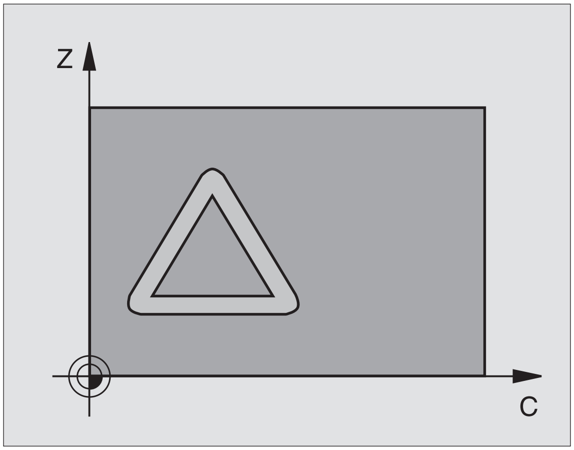
1 The TNC positions the tool over the cutter infeed point.

2 At the first plunging depth, the tool mills along the programmed slot wall at the milling feed rate Q12 while respecting the finishing allowance for the side.

3 At the end of the contour, the TNC moves the tool to the opposite wall and returns to the infeed point.

4 Steps 2 and 3 are repeated until the programmed milling depth Q1 is reached.

5 Then the tool moves to the setup clearance.



Before programming, note the following:

The memory capacity for programming an SL cycle is limited. For example, you can program up to 256 straight-line blocks in one SL cycle.

The algebraic sign for the cycle parameter DEPTH determines the working direction. If you program DEPTH = 0, the cycle will not be executed.

This cycle requires a center-cut end mill (ISO 1641).

The cylinder must be set up centered on the rotary table.

The tool axis must be perpendicular to the rotary table. If this is not the case, the TNC will generate an error message.

This cycle can also be used in a tilted working plane.

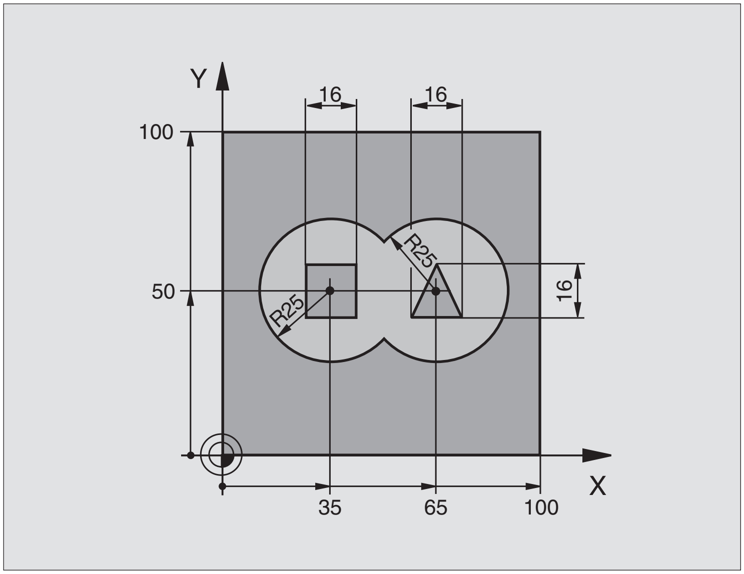
The TNC checks whether the compensated and non-compensated tool paths lie within the display range of the rotary axis, which is defined in Machine Parameter 810.x. If the error message “Contour programming error” is output, set MP 810.x = 0.

- **`Milling depth`** Q1 (incremental value): Distance between the cylindrical surface and the floor of the contour.
- **`Finishing allowance for side`** Q3 (incremental value): Finishing allowance in the plane of the unrolled cylindrical surface. This allowance is effective in the direction of the radius compensation.
- **`Set-up clearance`** Q6 (incremental value): Distance between the tool tip and the cylinder surface.
- **`Plunging depth`** Q10 (incremental value): Dimension by which the tool plunges in each infeed.
- **`Feed rate for plunging`** Q11: Traversing speed of the tool in the tool axis.
- **`Feed rate for milling`** Q12: Traversing speed of the tool in the working plane.
- **`Cylinder radius`** Q16: Radius of the cylinder on which the contour is to be machined.
- **`Dimension type ? ang./lin.`** Q17: The dimensions for the rotary axis of the subprogram are given either in degrees (0) or in mm/inches (1).
- **`Slot width`** Q20: Width of the slot to be machined.

Example: NC block

```
N63 G128 Q1=-8 Q3=+0 Q6=+0 Q10=+3 Q11=100
    Q12=350 Q16=25 Q17=0 Q20=12 *
```

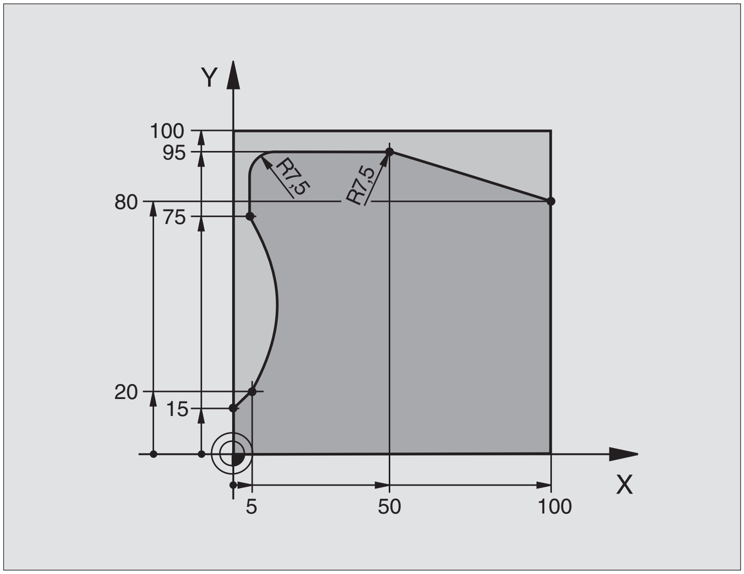
Example: Pilot drilling, roughing-out and finishing overlapping contours



%C21 G71 *	
N10 G30 G17 X+0 Y+0 Z-40 *	Define the workpiece blank
N20 G31 G90 X+100 Y+100 Z+0 *	
N30 G99 T1 L+0 R+6 *	Define tool: drill
N40 G99 T2 L+0 R+6 *	Define the tool for roughing/finishing
N50 T1 G17 S4000 *	Call the drilling tool
N60 G00 G40 G90 Z+250 *	Retract the tool
N70 G37 P01 1 P02 2 P03 3 P04 4 *	Define contour subprogram
N80 G120 Q1=-20 Q2=1 Q3=+0.5 Q4=+0.5	Define general machining parameters
Q5=+0 Q6=+2 Q7=+100 Q8=+0.1 Q9=-1 *	
N90 G121 Q10=+5 Q11=250 Q13=2 *	Cycle definition: Pilot drilling
N100 G79 M3 *	Cycle call: Pilot drilling
N110 Z+250 M6 *	Tool change
N120 T2 G17 S3000 *	Call the tool for roughing/finishing
N130 G122 Q10=+5 Q11=100 Q12=350 *	Cycle definition: Rough-out
N140 G79 M3 *	Cycle call: Rough-out
N150 G123 Q11=100 Q12=200 *	Cycle definition: Floor finishing
N160 G79 *	Cycle call: Floor finishing
N170 G124 Q9=+1 Q10=+5 Q11=100 Q12=400	Cycle definition: Side finishing
Q14=+0 *	

N180 G79 *	Cycle call: Side finishing
N190 G00 Z+250 M2 *	Retract in the tool axis, end program
N200 G98 L1 *	Contour subprogram 1: left pocket
N210 I+35 J+50 *	
N220 G01 G42 X+10 Y+50 *	
N230 G02 X+10 *	
N240 G98 L0 *	
N250 G98 L2 *	Contour subprogram 2: right pocket
N260 I+65 J+50 *	
N270 G01 G42 X+90 Y+50 *	
N280 G02 X+90 *	
N290 G98 L0 *	
N300 G98 L3 *	Contour subprogram 3: square left island
N310 G01 G41 X+27 Y+50 *	
N320 Y+58 *	
N330 X+43 *	
N340 Y+42 *	
N350 X+27 *	
N360 G98 L0 *	
N370 G98 L4 *	Contour subprogram 4: triangular right island
N380 G01 G41 X+65 Y+42 *	
N390 X+57 *	
N400 X+65 Y+58 *	
N410 X+73 Y+42 *	
N420 G98 L0 *	
N999999 %C21 G71 *	

Example: Contour train



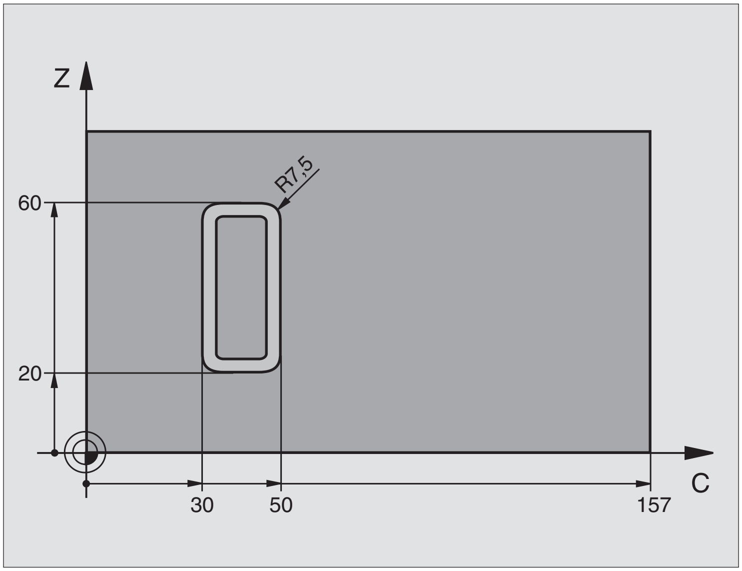
%C25 G71 *	
N10 G30 G17 X+0 Y+0 Z-40 *	Define the workpiece blank
N20 G31 G90 X+100 Y+100 Z+0 *	
N30 G99 T1 L+0 R+10 *	Define the tool
N50 T1 G17 S2000 *	Tool call
N60 G00 G40 G90 Z+250 *	Retract the tool
N70 G37 P01 1 *	Define contour subprogram
N80 G125 Q1=-20 Q3=+0 Q5=+0 Q7=+250	Define machining parameters
Q10=+5 Q11=100 Q12=200 Q15=+1 *	
N90 G79 M3 *	Call the cycle
N100 G00 G90 Z+250 M2 *	Retract in the tool axis, end program
N110 G98 L1 *	Contour subprogram
N120 G01 G41 X+0 Y+15 *	
N130 X+5 Y+20 *	
N140 G06 X+5 Y+75 *	
N150 G01 Y+95 *	
N160 G25 R7.5 *	
N170 X+50 *	
N180 G25 R7.5 *	
N190 X+100 Y+80 *	

N200 G98 L0 *	
N999999 %C25 G71 *	

Example: Cylinder surface

Note:

- Cylinder centered on rotary table
- Datum at center of rotary table



%C27 G71 *	
N10 G99 T1 L+0 R+3.5 *	Define the tool
N20 T1 G18 S2000 *	Call tool, tool axis is Y
N30 G00 G40 G90 Y+250 *	Retract the tool
N40 G37 P01 1 *	Define contour subprogram
N50 G127 Q1=-7 Q3=+0 Q6=+2 Q10=+4	Define machining parameters
Q11=100 Q12=250 Q16=25 *	
N60 C+0 M3 *	Pre-position rotary table
N70 G79 *	Call the cycle
N80 G00 G90 Y+250 M2 *	Retract in the tool axis, end program
N90 G98 L1 *	Contour subprogram
N100 G01 G41 C+91.72 Z+20 *	Data for the rotary axis are entered in degrees
N110 C+114.65 Z+20 *	Drawing dimensions are converted from mm to degrees (157 mm = 360°)
N120 G25 R7.5 *	
N130 G91 Z+40 *	
N140 G90 G25 R7.5 *	
N150 G91 C-45.86 *	
N160 G90 G25 R7.5 *	
N170 Z+20 *	
N180 G25 R7.5 *	

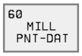
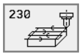
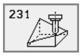
N190 C+91.72 *	
N200 G98 L0 *	
N999999 %C27 G71 *	

8.8 Cycles for Multipass Milling

Overview

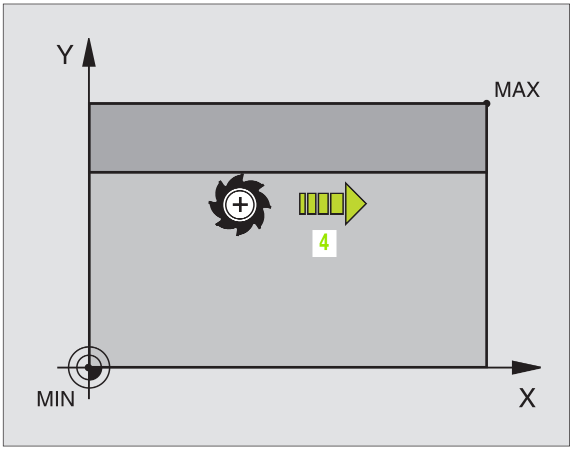
The TNC offers three cycles for machining the following surface types:

- Created by digitizing or with a CAD/CAM system
- Flat, rectangular surfaces
- Flat, oblique-angled surfaces
- Surfaces that are inclined in any way
- Twisted surfaces

Cycle	Soft key
G60 RUN DIGITIZED DATA For multipass milling of digitized surface data in several infeeds	
G230 MULTIPASS MILLING For flat rectangular surfaces	
G231 RULED SURFACE For oblique, inclined or twisted surfaces	

RUN DIGITIZED DATA (Cycle G60, not TNC 410)

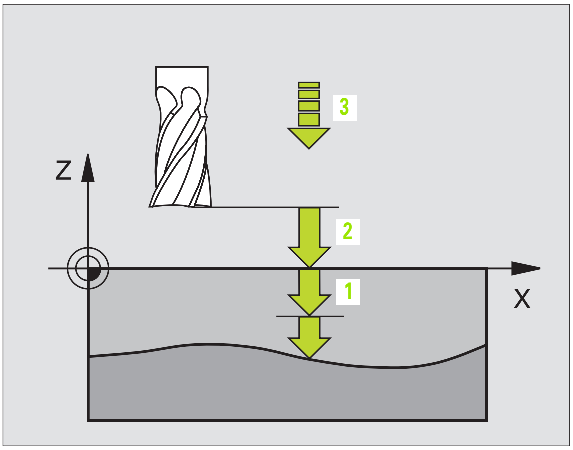
1 From the current position, the TNC positions the tool in rapid traverse in the tool axis to the set-up clearance above the MAX point that you have programmed in the cycle.
2 The tool then moves in rapid traverse in the working plane to the MIN point you have programmed in the cycle.
3 From this point, the tool advances to the first contour point at the feed rate for plunging.
4 The TNC subsequently processes all points that are stored in the digitizing data file at the feed rate for milling. If necessary, the TNC retracts the tool between machining operations to set-up clearance if specific areas are to be left unmachined.
5 At the end of the cycle, the tool is retracted in rapid traverse to set-up clearance.



Before programming, note the following:

You can use Cycle G60 to run digitizing data and PNT files.

If you want to run PNT files in which no tool axis coordinate is programmed, the milling depth is derived from the programmed MIN point in the tool axis.



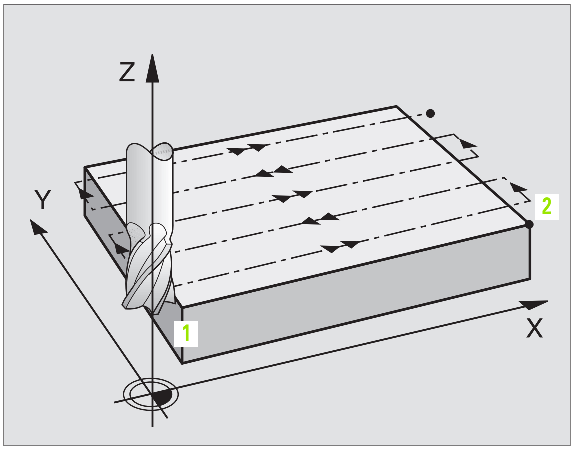
60 MILL PNT-DAT

- **PGM Name digitizing data**: Enter the name of the file in which the digitizing data is stored. If the file is not stored in the current directory, enter the complete path. If you wish to execute a point table, enter also the file type .PNT.
- **Min. point of range**: Lowest coordinates (X, Y and Z coordinates) in the range to be milled.
- **Max. point of range**: Highest coordinates (X, Y and Z coordinates) in the range to be milled.
- **Set-up clearance 1** (incremental value): Distance between tool tip and workpiece surface for tool movements in rapid traverse.
- **Plunging depth 2** (incremental value): Infeed per cut.
- **Feed rate for plunging 3**: Traversing speed of the tool in mm/min during penetration.
- **Feed rate for milling 4**: Traversing speed of the tool in mm/min while milling.
- **Miscellaneous function M**: Optional entry of a miscellaneous function, for example M13.

Example: NC block

```
N64 G60 P01 BSP.I P02 X+0 P03 Y+0
    P04 Z-20 P05 X+100 P06 Y+100 P07 Z+0
    P08 2 P09 +5 P10 100 P11 350
    P12 M13 *
```

MULTIPLASS MILLING (Cycle G230)

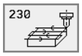


1 From the current position in the working plane, the TNC positions the tool at rapid traverse to the starting point **1**; the TNC moves the tool by its radius to the left and upward.
2 The tool then moves in rapid traverse in the tool axis to set-up clearance. From there it approaches the programmed starting position in the tool axis at the feed rate for plunging.
3 The tool then moves as the programmed feed rate for milling to the end point **2**. The TNC calculates the end point from the programmed starting point, the program length, and the tool radius.
4 The TNC offsets the tool to the starting point in the next pass at the stepover feed rate. The offset is calculated from the programmed width and the number of cuts.
5 The tool then returns in the negative direction of the first axis.
6 Multipass milling is repeated until the programmed surface has been completed.
7 At the end of the cycle, the tool is retracted in rapid traverse to set-up clearance.

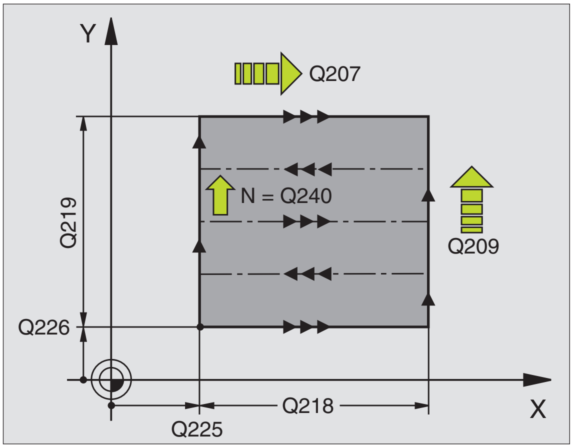
Before programming, note the following:

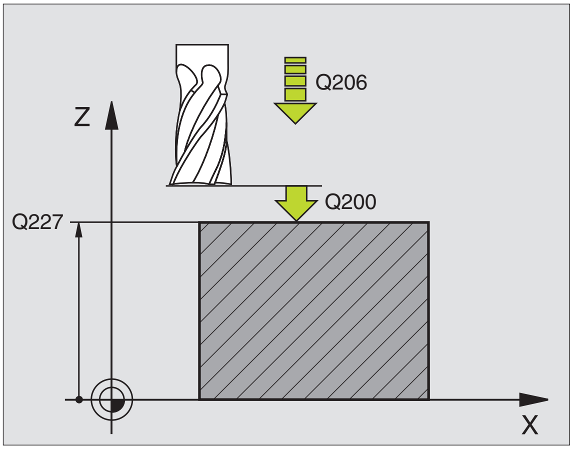
From the current position, the TNC positions the tool at the starting point 1, first in the working plane and then in the tool axis.

Pre-position the tool in such a way that no collision between tool and clamping devices can occur.



- **`Starting point in 1st axis`** Q225 (absolute value): Minimum point coordinate of the surface to be multipass-milled in the reference axis of the working plane.
- **`Starting point in 2nd axis`** Q226 (absolute value): Minimum-point coordinate of the surface to be multipass-milled in the minor axis of the working plane.
- **`Starting point in 3rd axis`** Q227 (absolute value): Height in the spindle axis at which multipass-milling is carried out.
- **`First side length`** Q218 (incremental value): Length of the surface to be multipass-milled in the reference axis of the working plane, referenced to the starting point in 1st axis
- **`Second side length`** Q219 (incremental value): Length of the surface to be multipass-milled in the minor axis of the working plane, referenced to the starting point in 2nd axis
- **`Number of cuts`** Q240: Number of passes to be made over the width.
- **`Feed rate for plunging`** Q206: Traversing speed of the tool in mm/min when moving from set-up clearance to the milling depth.
- **`Feed rate for milling`** Q207: Traversing speed of the tool in mm/min while milling.
- **`Stepover feed rate`** Q209: Traversing speed of the tool in mm/min when moving to the next pass. If you are moving the tool transversely in the material, enter Q209 to be smaller than Q207. If you are moving it transversely in the open, Q209 may be greater than Q207.
- **`Set-up clearance`** Q200 (incremental value): Distance between tool tip and milling depth for positioning at the start and end of the cycle.

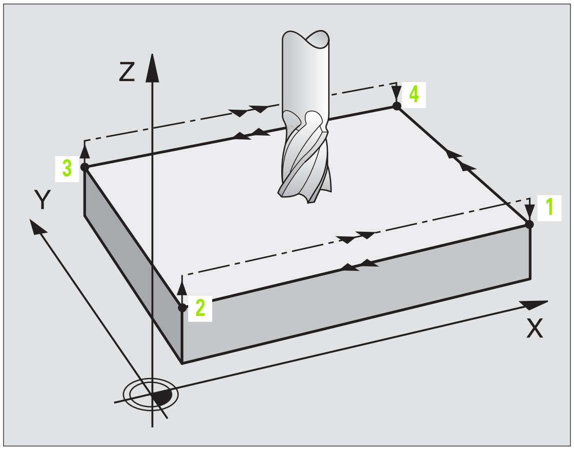




Example: NC block

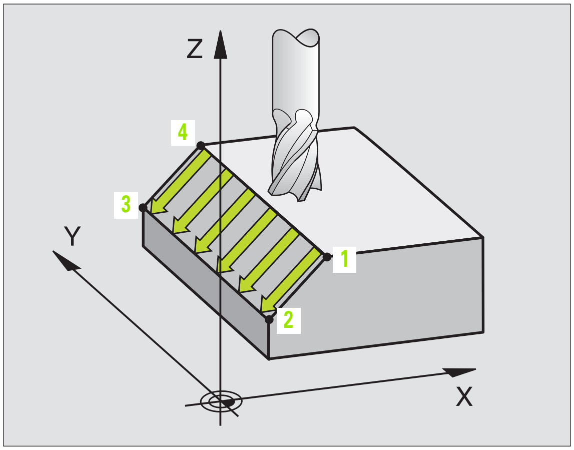
```
N71 G230 Q225=+10 Q226=+12 Q227=+2.5
    Q218=150 Q219=75 Q240=25 Q206=150
    Q207=500 Q209=200 Q200=2 *
```

RULED SURFACE (Cycle G231)



1 From the current position, the TNC positions the tool in a linear 3-D movement to the starting point **1**.
2 The tool subsequently advances to the stopping point **2** at the feed rate for milling.
3 From this point, the tool moves at rapid traverse by the tool diameter in the positive tool axis direction, and then back to starting point **1**.
4 At the starting point **1** the TNC moves the tool back to the last traversed Z value.
5 Then the TNC moves the tool in all three axes from point **1** in the direction of point **4** to the next line.
6 From this point, the tool moves to the stopping point on this pass. The TNC calculates the end point from point **2** and a movement in the direction of point **3**.
7 Multipass milling is repeated until the programmed surface has been completed.
8 At the end of the cycle, the tool is positioned above the highest programmed point in the tool axis, offset by the tool diameter.

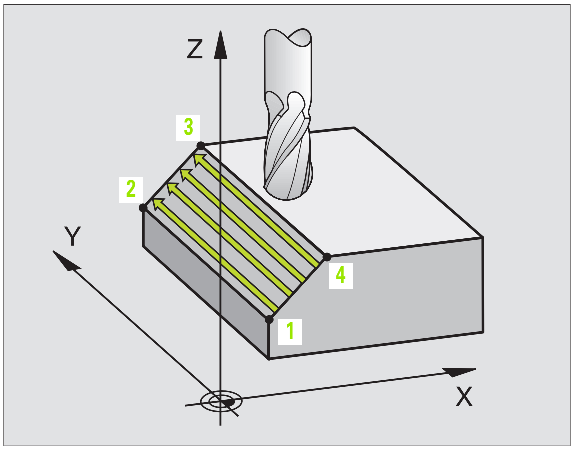
Cutting motion



The starting point, and therefore the milling direction, is selectable because the TNC always moves from point **1** to point **2** and in the total movement from point **1** / **2** to point **3** / **4**. You can program point **1** at any corner of the surface to be machined.

If you are using an end mill for the machining operation, you can optimize the surface finish in the following ways:

- A shaping cut (spindle axis coordinate of point **1** greater than spindle-axis coordinate of point **2**) for slightly inclined surfaces.
- A drawing cut (spindle axis coordinate of point **1** smaller than spindle-axis coordinate of point **2**) for steep surfaces.
- When milling twisted surfaces, program the main cutting direction (from point **1** to point **2**) parallel to the direction of the steeper inclination.



If you are using a spherical cutter for the machining operation, you can optimize the surface finish in the following way:

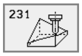
- When milling twisted surfaces, program the main cutting direction (from point **1** to point **2**) perpendicular to the direction of the steepest inclination.

Before programming, note the following:

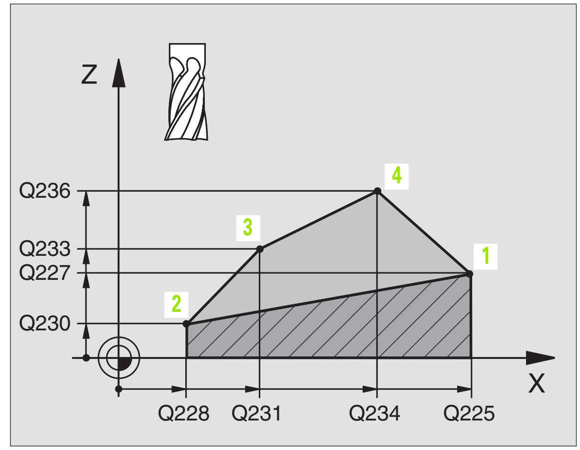
The TNC positions the tool from the current position in a linear 3-D movement to the starting point **1**. Pre-position the tool in such a way that no collision between tool and clamping devices can occur.

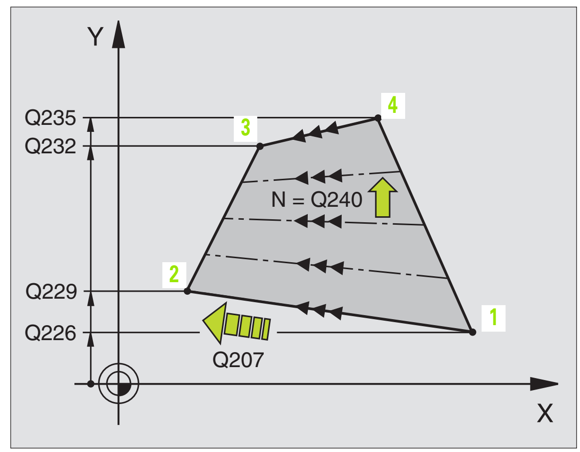
The TNC moves the tool with radius compensation **G40** to the programmed positions.

If required, use a center-cut end mill (ISO 1641).



- **`Starting point in 1st axis`** Q225 (absolute value): Starting point coordinate of the surface to be multipass-milled in the reference axis of the working plane.
- **`Starting point in 2nd axis`** Q226 (absolute value): Starting point coordinate of the surface to be multipass-milled in the minor axis of the working plane.
- **`Starting point in 3rd axis`** Q227 (absolute value): Starting point coordinate of the surface to be multipass-milled in the tool axis.
- **`2nd point in 1st axis`** Q228 (absolute value): Stopping point coordinate of the surface to be multipass milled in the reference axis of the working plane
- **`2nd point in 2nd axis`** Q229 (absolute value): Stopping point coordinate of the surface to be multipass milled in the minor axis of the working plane
- **`2nd point in 3rd axis`** Q230 (absolute value): Stopping point coordinate of the surface to be multipass milled in the tool axis
- **`3rd point in 1st axis`** Q231 (absolute value): Coordinate of point 3 in the reference axis of the working plane
- **`3rd point in 2nd axis`** Q232 (absolute value): Coordinate of point 3 in the minor axis of the working plane
- **`3rd point in 3rd axis`** Q233 (absolute value): Coordinate of point 3 in the tool axis
- **`4th point in 1st axis`** Q234 (absolute value): Coordinate of point 4 in the reference axis of the working plane
- **`4th point in 2nd axis`** Q235 (absolute value): Coordinate of point 4 in the minor axis of the working plane
- **`4th point in 3rd axis`** Q236 (absolute value): Coordinate of point 4 in the tool axis
- **`Number of cuts`** Q240: Number of passes to be made between points 1 and 4, 2 and 3.
- **`Feed rate for milling`** Q207: Traversing speed of the tool in mm/min while milling. The TNC performs the first step at half the programmed feed rate.

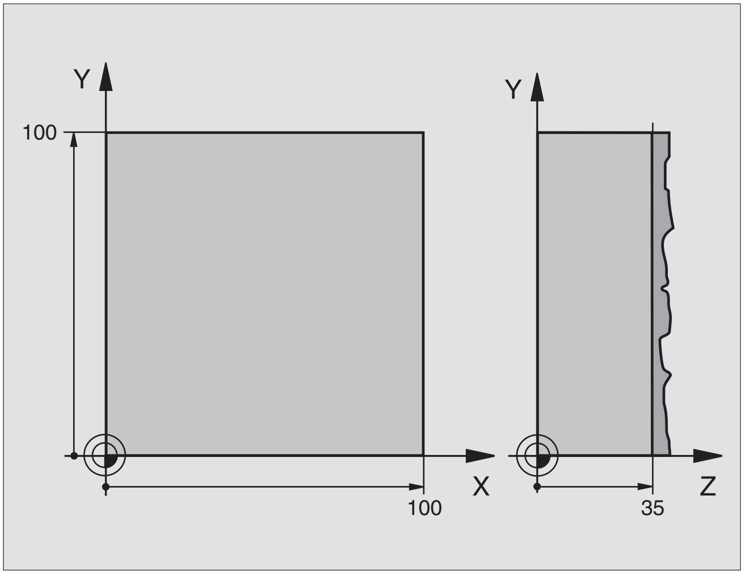




Example: NC blocks

```
N72 G231 Q225=+0 Q226=+5 Q227=-2
    Q228=+100 Q229=+15 Q230=+5 Q231=+15
    Q232=+125 Q233=+25 Q234=+15 Q235=+125
    Q236=+25 Q240=40 Q207=500 *
```

Example: Multipass milling

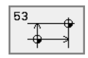
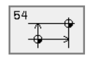
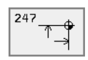
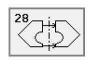
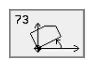
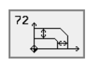
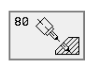


`%C230 G71`	
`N10 G30 G17 X+0 Y+0 Z+0 *`	Define the workpiece blank
`N20 G31 G90 X+100 Y+100 Z+40 *`	
`N30 G99 T1 L+0 R+5 *`	Define the tool
`N40 T1 G17 S3500 *`	Tool call
`N50 G00 G40 G90 Z+250 *`	Retract the tool
`N60 G230 Q225=+0 Q226=+0 Q227=+35`	Cycle definition: MULTIPASS MILLING
`Q218=100 Q219=100 Q240=25 Q206=250`	
`Q207=400 Q209=150 Q200=2 *`	
`N70 X-25 Y+0 M03 *`	Pre-position near the starting point
`N80 G79 *`	Call the cycle
`N90 G00 G40 Z+250 M02 *`	Retract in the tool axis, end program
`N999999 %C230 G71 *`	

8.9 Coordinate Transformation Cycles

Overview

Once a contour has been programmed, you can position it on the workpiece at various locations and in different sizes through the use of coordinate transformations. The TNC provides the following coordinate transformation cycles:

Cycle	Soft key
G53/G54 DATUM SHIFT For shifting contours directly within the program or from datum tables	 
G247 DATUM SETTING Datum setting during program run (not TNC 410)	
G28 MIRROR IMAGE Mirroring contours	
G73 ROTATION For rotating contours in the working plane	
G72 SCALING FACTOR For increasing or reducing the size of contours	
G80 WORKING PLANE For executing machining operations in a tilted coordinate system for machines with tilting heads and/or rotary tables (not TNC 410)	

Effect of coordinate transformations

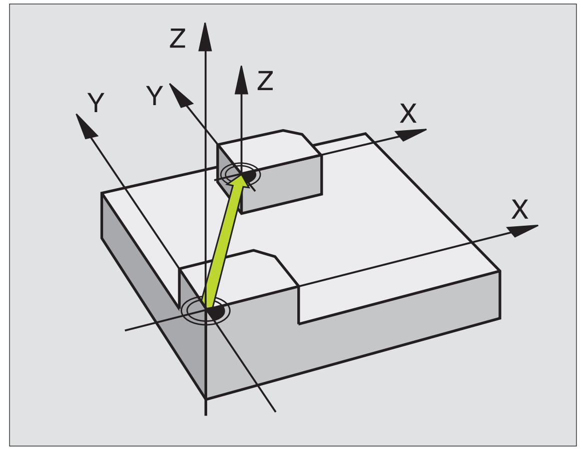
Beginning of effect: A coordinate transformation becomes effective as soon as it is defined—it is not called. It remains in effect until it is changed or canceled.

To cancel coordinate transformations:

- Define cycles for basic behavior with a new value, such as scaling factor 1.0.
- Execute a miscellaneous function M02, M30, or an N999999 %... block (depending on Machine Parameter 7300).
- Select a new program.
- Program miscellaneous function M142 Erasing modal program information.

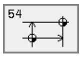
DATUM SHIFT (Cycle G54)

A DATUM SHIFT allows machining operations to be repeated at various locations on the workpiece.



Effect

When the DATUM SHIFT cycle is defined, all coordinate data is based on the new datum. The TNC displays the datum shift in each axis in the additional status display. Input of rotary axes is also permitted.

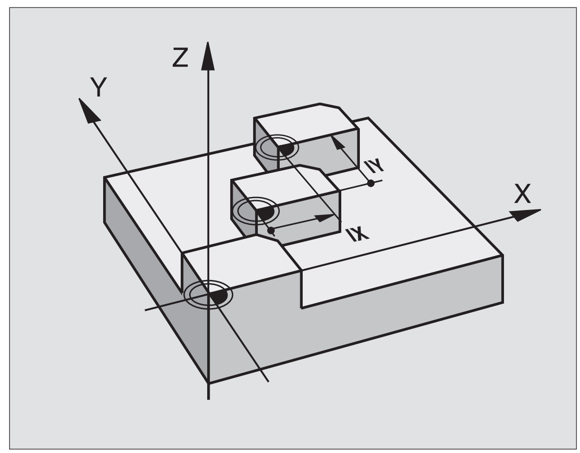


- **`Datum shift:`** Enter the coordinates of the new datum. Absolute values are referenced to the manually set workpiece datum. Incremental values are always referenced to the datum which was last valid—this can be a datum which has already been shifted.

Additionally with TNC 410:

REF

- **`REF`**: Press the REF soft key to reference the programmed datum to the machine datum. In this case the TNC indicates the first cycle block with **`REF`**



Cancellation

A datum shift is canceled by entering the datum shift coordinates X=0, Y=0 and Z=0.

Graphics (not TNC 410)

If you program a new workpiece blank after a datum shift, you can use Machine Parameter 7310 to determine whether the blank is referenced to the current datum or to the original datum. Referencing a new BLK FORM to the current datum enables you to display each part in a program in which several pallets are machined.

Status displays

- The actual position values are referenced to the active (shifted) datum.
- All of the position values shown in the additional status display are referenced to the manually set datum.

Example: NC blocks

```
N72 G54 G90 X+25 Y-12.5 Z+100 *
...
N78 G54 G90 REF X+25 Y-12.5 Z+100 *
```

DATUM SHIFT with datum tables (Cycle G53)

Datums from a datum table can be referenced either to the current datum or to the machine datum (depending on machine parameter 7475).

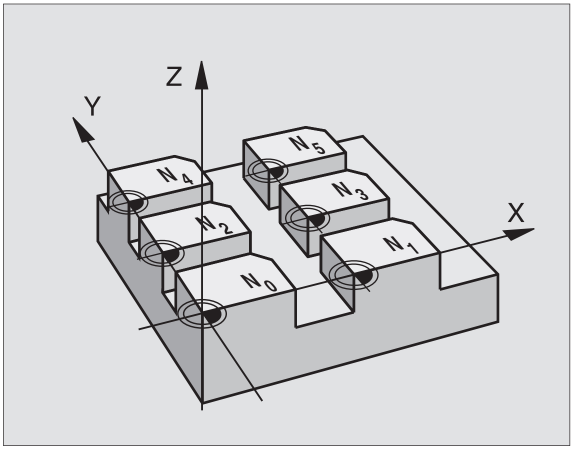
The coordinate values from datum tables are only effective with absolute coordinate values.

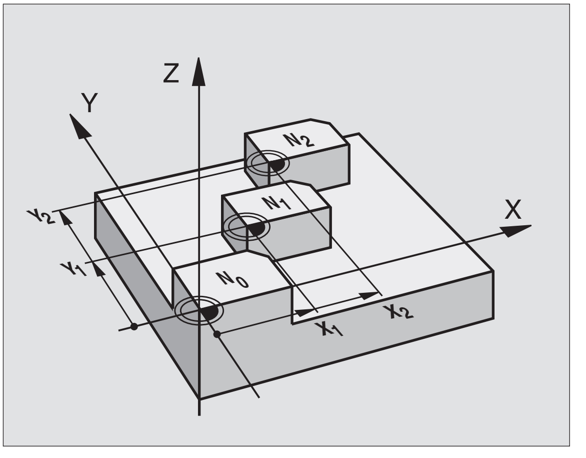
Not available with TNC 410:

To use a datum table, you must activate the desired datum table before the test run or program run (this applies also for the programming graphics):

- Use the file management to select the desired table for a test run in the **`Test Run`** operating mode: The table receives the status S.
- Use the file management in a program run mode to select the desired table for a program run: The table receives the status M.

- New lines can only be inserted at the end of the table.
- Working with only one datum table ensures that the correct datum is activated in the program run modes of operation.



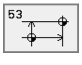


Function

Datum tables are used for

- frequently recurring machining sequences at various locations on the workpiece
- frequent use of the same datum shift

Within a program, you can either program datum points directly in the cycle definition or call them from a datum table.



▶ **`Datum shift:`** Enter the number of the datum from the datum table or a Q parameter. If you enter a Q parameter, the TNC activates the datum number found in the Q parameter.

Cancellation

- Call a datum shift to the coordinates X=0; Y=0 etc. from the datum table.
- Execute a datum shift to the coordinates X=0; Y=0 etc. directly with a cycle definition.

Example: NC blocks

```
N72 G53 P01 12 *
```

Editing a datum table with the TNC 410

Select the datum table in the **`Programming and Editing`** mode of operation.

PGM MGT

- To call the file manager, press the PGM MGT key, see “File Management: Fundamentals,” page 43.
- To select an already existing datum table, move the highlight to the desired table and confirm with the ENT key
- To open a new datum table, enter a new file name and confirm with the ENT key. Press the “.D” soft key to open the table.

Editing a datum table with TNC 426, TNC 430

Select the datum table in the **`Programming and Editing`** mode of operation.

PGM MGT

- To call the file manager, press the PGM MGT key; see “File Management: Fundamentals,” page 43.
- Display the datum tables: Press the soft keys SELECT TYPE and SHOW .D.
- Select the desired table or enter a new file name.
- Edit the file. The soft-key row comprises the following functions for editing:

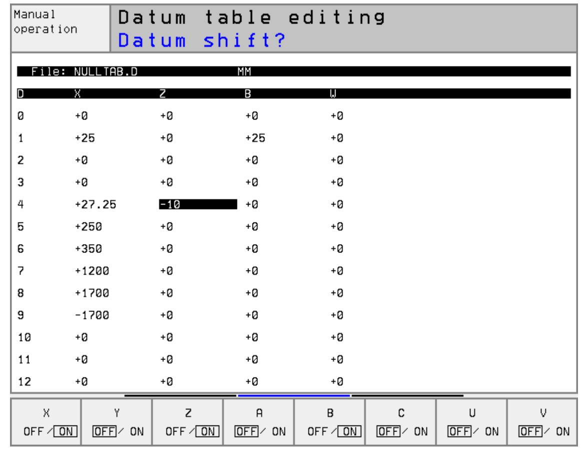
Function	Soft key
Select beginning of table	BEGIN ⇧
Select end of table	END ⇩
Go to previous page	PAGE ⇧
Go to next page	PAGE ⇩
Insert line (only possible at end of table)	INSERT LINE
Delete line	DELETE LINE
Confirm the entered line and go to the beginning of the next line (not TNC 410)	NEXT LINE
Add the entered number of lines (reference points) to the end of the table	APPEND N LINES
Move the highlight one column to the left (not TNC 410)	WORT ⇦
Move the highlight one column tothe right (not TNC 410)	WORT ⇨

> With the function “Actual position capture” the TNC stores the positions of that axis in the header of the table which is above the marked value (not TNC 410).

Configuring datum tables (not TNC 410)

On the second and third soft-key rows you can define for each datum table the axes for which you wish to set the datums. In the standard setting all of the axes are active. If you wish to exclude an axis, set the corresponding soft key to OFF. The TNC then deletes that column from the datum table.

If you do not wish to define a datum table for an active axis, press the NO ENT key. The TNC then enters a dash in the corresponding column.



To leave a datum table

Select a different type of file in file management and choose the desired file.

Status displays

If datums in the table are referenced to the machine datum, then:

- The actual position values are referenced to the active (shifted) datum.
- All of the position values shown in the additional status display are referenced to the machine datum, whereby the TNC accounts for the manually set datum.

Activating a datum table for program run (TNC 410)

With the TNC 410 you must use the function %:TAB: in the NC program to select the datum table from which the TNC is to take the datums:

PGM CALL

- To select the function for program call, press the PGM CALL key.
- Press the DATUM TABLE soft key.
- Enter the name of the datum table and confirm your entry with the END key.

Example NC block:

```
N72 %:TAB: "NAMES"*
```

Activating a datum table for program run (TNC 426, TNC 430

With the TNC 426, TNC 430 you must manually activate the datum table in a program run mode of operation:

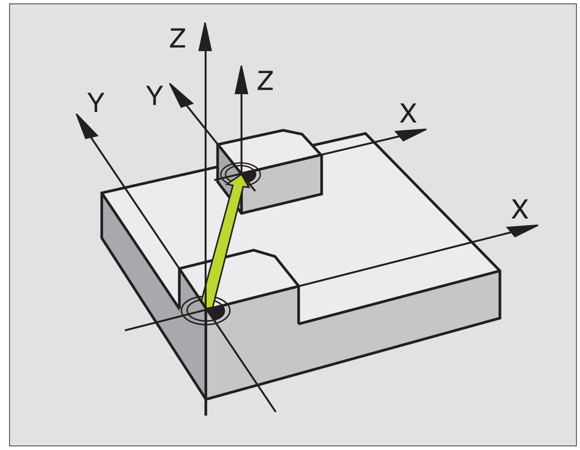
- Select a program run mode, e.g., Program Run, Full Sequence

PGM MGT

- To call the file manager, press the PGM MGT key; see “File Management: Fundamentals,” page 43.
- To select an already existing datum table, move the highlight to the desired table and confirm with the ENT key. The TNC indicates with M the selected table in the status field.

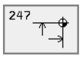
DATUM SETTING (Cycle G247, not TNC 410)

With the Cycle DATUM SETTING, you can activate a datum defined in a datum table as the new datum.



Effect

After a DATUM SETTING cycle definition, all of the coordinate inputs and datum shifts (absolute and incremental) are referenced to the new datum. Setting datums for rotary axes is also possible.



- **`Number for datum?`**: Enter the number of the datum in the datum table.

Cancellation

You can reactivate the last datum set in the Manual mode by entering the miscellaneous function M104.

The TNC only sets the datum for those axes which are active in the datum table. An axis displayed as a column in the datum table, but not existing on the TNC, will cause an error message.

Cycle G247 always interprets the values saved in the datum table as coordinates referenced to the machine datum. Machine parameter 7475 has no influence on this.

When using Cycle G247, you cannot use the block scan function for mid-program startup.

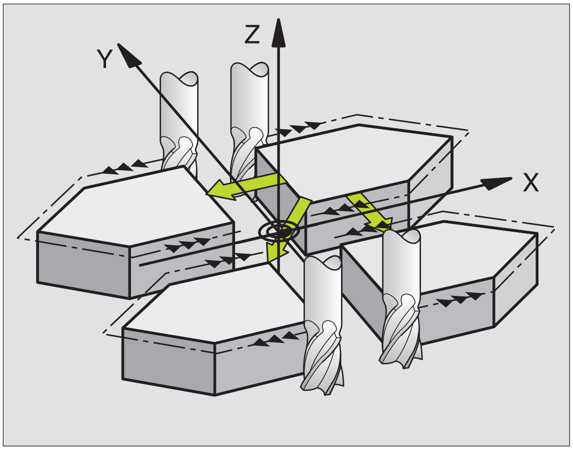
Cycle G247 is not functional in Test Run mode.

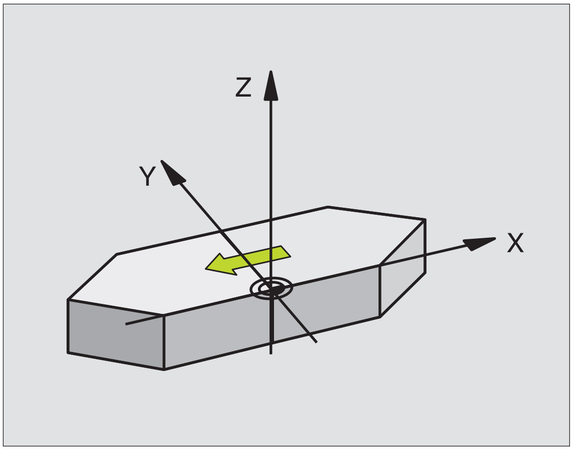
Example: NC block

```
N13 G247 Q339=4 *
```

MIRROR IMAGE (Cycle G28)

The TNC can machine the mirror image of a contour in the working plane.





Effect

The mirror image cycle becomes effective as soon as it is defined in the program. It is also effective in the Positioning with MDI mode of operation. The active mirrored axes are shown in the additional status display.

- If you mirror only one axis, the machining direction of the tool is reversed (except in fixed cycles).
- If you mirror two axes, the machining direction remains the same.

The result of the mirror image depends on the location of the datum:

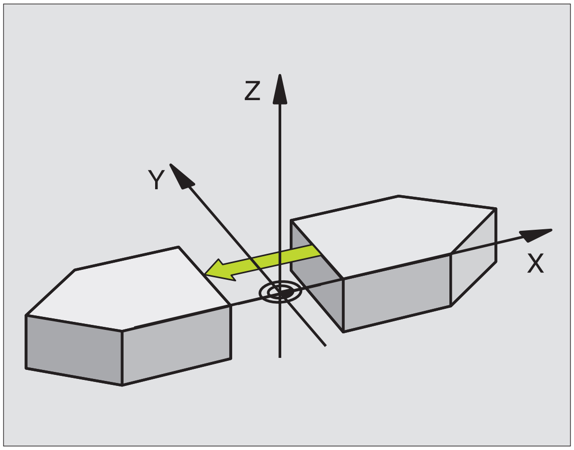
- If the datum lies on the contour to be mirrored, the element simply flips over.
- If the datum lies outside the contour to be mirrored, the element also "jumps" to another location.

> If you mirror only one axis, the machining direction is reversed for the new machining cycles (cycles 2xx). The machining direction remains the same for older machining cycles, such as Cycle 4 POCKET MILLING.

▶ **`Mirrored axis?`**: Enter the axes to be mirrored. You can mirror all axes, including rotary axes, except for the spindle axis and its auxiliary axes. You can enter up to three axes.

Reset

Program the MIRROR IMAGE cycle once again with NO ENT.

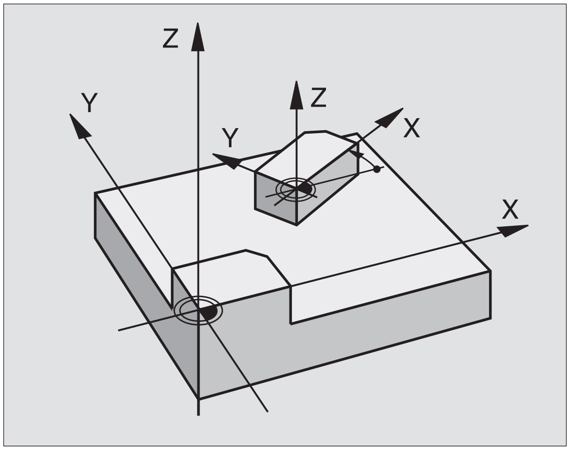


Example: NC block

```
N72 G28 X Y *
```

ROTATION (Cycle G73)

The TNC can rotate the coordinate system about the active datum in the working plane within a program.



Effect

The ROTATION cycle becomes effective as soon as it is defined in the program. It is also effective in the Positioning with MDI mode of operation. The active rotation angle is shown in the additional status display.

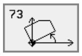
Reference axis for the rotation angle:

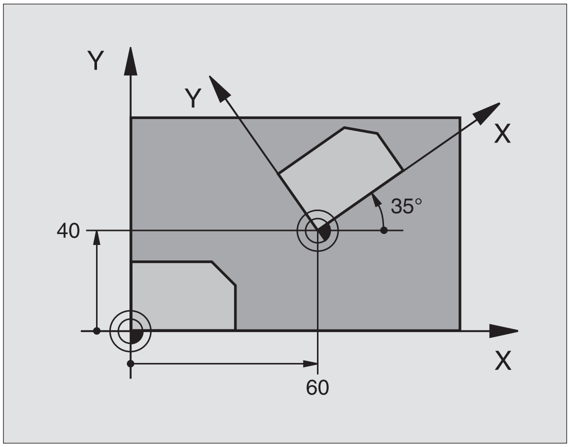
- X/Y plane X axis
- Y/Z plane Y axis
- Z/X plane Z axis

Before programming, note the following:

An active radius compensation is canceled by defining Cycle **G73** and must therefore be reprogrammed, if necessary.

After defining Cycle **G73**, you must move both axes of the working plane to activate rotation for all axes.





- **Rotation:** Enter the rotation angle in degrees (°). Input range: -360° to +360° (absolute G90 before H or incremental G91 before H).

Cancellation

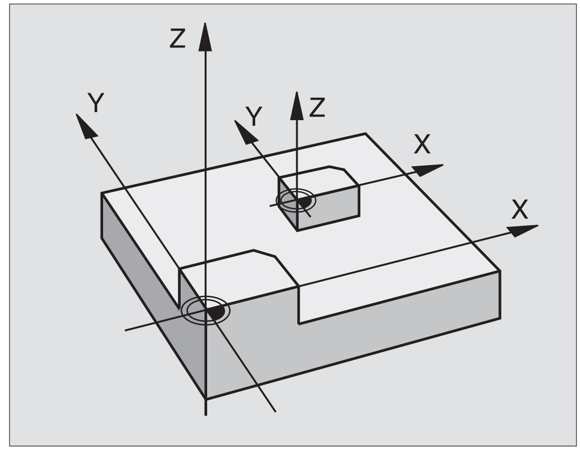
Program the ROTATION cycle once again with a rotation angle of 0°.

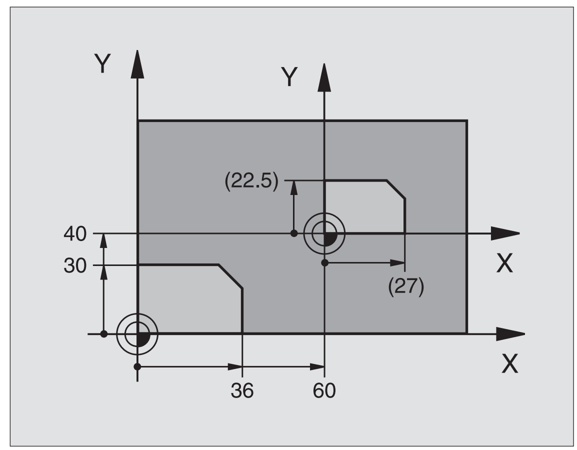
Example: NC block

```
N72 G73 G90 H+25 *
```

SCALING FACTOR (Cycle G72)

The TNC can increase or reduce the size of contours within a program, enabling you to program shrinkage and oversize allowances.





Effect

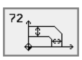
The SCALING FACTOR becomes effective as soon as it is defined in the program. It is also effective in the Positioning with MDI mode of operation. The active scaling factor is shown in the additional status display.

The scaling factor affects

- in the working plane, or on all three coordinate axes at the same time (depending on machine parameter 7410)
- dimensions in cycles
- the parallel axes U,V,W

Prerequisite

It is advisable to set the datum to an edge or a corner of the contour before enlarging or reducing the contour.



▶ **`Scaling factor?`**: Enter the scaling factor F. The TNC multiplies the coordinates and radii by the F factor (as described under “Activation” above).

Enlargement: F greater than 1 (up to 99.999 999)

Reduction: F less than 1 (down to 0.000 001)

Cancellation

Program the SCALING FACTOR cycle once again with a scaling factor of 1 for the same axis.

Example: NC blocks

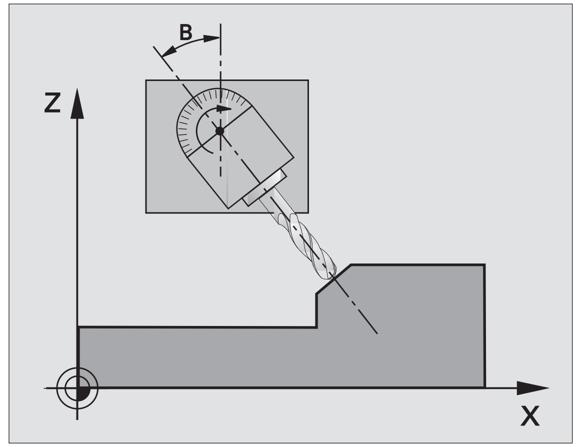
```
N72 G72 F0.750000 *
```

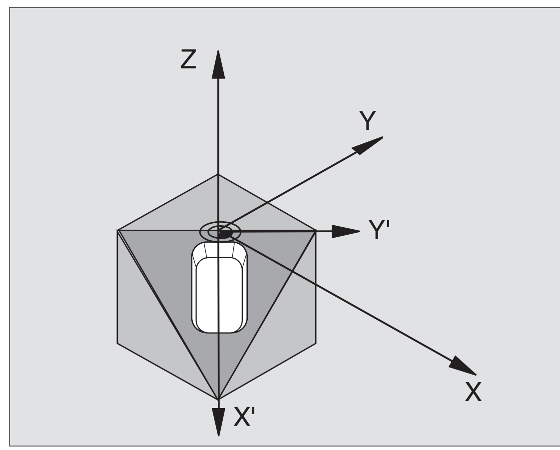
WORKING PLANE (Cycle G80, not TNC 410)

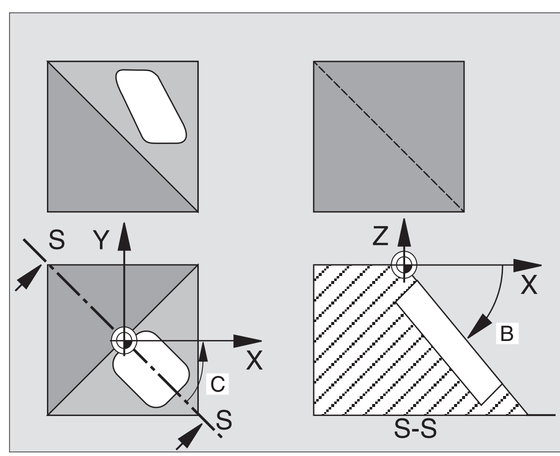
The functions for tilting the working plane are interfaced to the TNC and the machine tool by the machine tool builder. With some swivel heads and tilting tables, the machine tool builder determines whether the entered angles are interpreted as coordinates of the tilt axes or as mathematical angles of a tilted plane. Refer to your machine manual.

The working plane is always tilted around the active datum.

For fundamentals, see “Tilting the Working Plane (not TNC 410),” page 26. Please read this section completely.







Effect

In Cycle **G80** you define the position of the working plane—i.e. the position of the tool axis referenced to the machine coordinate system—by entering tilt angles. There are two ways to determine the position of the working plane:

- Enter the position of the tilting axes directly.
- Describe the position of the working plane using up to 3 rotations (spatial angle) of the **machine-referenced** coordinate system. The required spatial angle can be calculated by cutting a perpendicular line through the tilted working plane and considering it from the axis around which you wish to tilt. With two spatial angles, every tool position in space can be defined exactly.

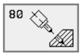
Note that the position of the tilted coordinate system, and therefore also all movement in the tilted system, are dependent on your description of the tilted plane.

If you program the position of the working plane via spatial angles, the TNC will calculate the required angle positions of the tilted axes automatically and will store these in the parameters Q120 (A axis) to Q122 (C axis). If two solutions are possible, the TNC will choose the shorter path from the zero position of the rotary axes.

The axes are always rotated in the same sequence for calculating the tilt of the plane: The TNC first rotates the A axis, then the B axis, and finally the C axis.

Cycle 19 becomes effective as soon as it is defined in the program. As soon as you move an axis in the tilted system, the compensation for this specific axis is activated. You have to move all axes to activate compensation for all axes.

If you set the function TILTING program run to ACTIVE in the Manual Operation mode (see "Tilting the Working Plane (not TNC 410)," page 26), the angular value entered in this menu is overwritten by Cycle **G80** WORKING PLANE.



- **Tilt axis and tilt angle?**: The axes of rotation together with the associated tilt angles. The rotary axes A, B and C are programmed using soft keys.

If the TNC automatically positions the rotary axes, you can enter the following parameters:

- **Feed rate ? F=**: Traverse speed of the rotary axis during automatic positioning.
- **Set-up clearance ?**(incremental value): The TNC positions the tilting head so that the position that results from the extension of the tool by the set-up clearance does not change relative to the workpiece.

Cancellation

To cancel the tilt angle, redefine the WORKING PLANE cycle and enter an angular value of 0° for all axes of rotation. You must then program the WORKING PLANE cycle again, without defining an axis, to disable the function.

Position the axis of rotation

The machine tool builder determines whether Cycle **G80** positions the axes of rotation automatically or whether they must be pre-positioned in the program. Refer to your machine manual.

If the axes are positioned automatically in Cycle **G80:**

- The TNC can position only controlled axes.
- In order for the tilted axes to be positioned, you must enter a feed rate and a set-up clearance in addition to the tilting angles, during cycle definition.
- You can use only preset tools (with the full tool length defined in the **G99** block or in the tool table).
- The position of the tool tip as referenced to the workpiece surface remains nearly unchanged after tilting.
- The TNC tilts the working plane at the last programmed feed rate. The maximum feed rate that can be reached depends on the complexity of the swivel head or tilting table.

If the axes are not positioned automatically in Cycle **G80,** position them before defining the cycle, for example with a G01 block.

Example NC blocks:

N50 G00 G40 Z+100 *	
N60 X+25 Y+10 *	
N70 G01 A+15 F1000 *	Position the axis of rotation
N80 G80 A+15 *	Define the angle for calculation of the compensation
N90 G00 G40 Z+80 *	Activate compensation for the tool axis
N100 X-7.5 Y-10 *	Activate compensation for the working plane

Position display in the tilted system

On activation of Cycle **G80,** the displayed positions (**ACTL** and **NOML**) and the datum indicated in the additional status display are referenced to the tilted coordinate system. The positions displayed immediately after cycle definition may not be the same as the coordinates of the last programmed position before Cycle **G80.**

Workspace monitoring

The TNC monitors only those axes in the tilted coordinate system that are moved. If necessary, the TNC outputs an error message.

Positioning in a tilted coordinate system

With the miscellaneous function M130 you can move the tool, while the coordinate system is tilted, to positions that are referenced to the non-tilted coordinate system see “Miscellaneous Functions for Coordinate Data,” page 150.

Positioning movements with straight lines that are referenced to the machine coordinate system (blocks with M91 or M92), can also be executed in a tilted working plane. Constraints:

- Positioning is without length compensation.
- Positioning is without machine geometry compensation.
- Tool radius compensation is not permitted.

Combining coordinate transformation cycles

When combining coordinate transformation cycles, always make sure the working plane is swiveled around the active datum. You can program a datum shift before activating Cycle **G80.** In this case, you are shifting the "machine-based coordinate system."

If you program a datum shift after having activated Cycle **G80,** you are shifting the "tilted coordinate system."

Important: When resetting the cycles, use the reverse sequence used for defining them:

1st: Activate the datum shift.
2nd: Activate tilting function.
3rd: Activate rotation.
...
Machining
...
1st: Reset the rotation.
2nd: Reset the tilting function.
3rd: Reset the datum shift.

Automatic workpiece measurement in the tilted system

The TNC measuring cycles enable you to have the TNC measure a workpiece in a tilted system automatically. The TNC stores the measured data in Q parameters for further processing (for example, for printout).

Procedure for working with Cycle G80 WORKING PLANE

1 Write the program

- Define the tool (not required, when TOOL.T is active), and enter the full tool length.
- Call the tool.
- Retract the tool in the tool axis to a position where there is no danger of collision with the workpiece (clamping devices) during tilting.
- If required, position the tilt axis or axes with a **G01** block to the appropriate angular value(s) (depending on a machine parameter).
- Activate datum shift if required.
- Define Cycle **G80** WORKING PLANE; enter the angular values for the tilt axes.
- Traverse all main axes (X, Y, Z) to activate compensation.
- Write the program as if the machining process were to be executed in a non-tilted plane.
- If required, define Cycle **G80** WORKING PLANE with other angular values to execute machining in a different axis position. In this case, it is not necessary to reset Cycle **G80.** You can define the new angular values directly.
- Reset Cycle **G80** WORKING PLANE; program 0° for all tilt axes.
- Disable the WORKING PLANE function; redefine Cycle **G80,** without defining an axis.
- Reset datum shift if required.
- Position the tilt axes to the 0° position, if required.

2 Clamp the workpiece

3 Preparations in the operating mode Positioning with Manual Data Input (MDI)

Pre-position the tilt axis/axes to the corresponding angular value(s) for setting the datum. The angular value depends on the selected reference plane on the workpiece.

4 Preparations in the operating mode Manual Operation

Use the 3D-ROT soft key to set the function TILT WORKING PLANE to ACTIVE in the Manual Operating mode. Enter the angular values for the tilt axes into the menu if the axes are not controlled.

If the axes are not controlled, the angular values entered in the menu must correspond to the actual position(s) of the tilted axis or axes, respectively. The TNC will otherwise calculate a wrong datum.

5 Set the datum

- Manually by touching the workpiece with the tool in the untilted coordinate system see "Datum Setting (Without a 3-D Touch Probe)," page 24
- Automatically by using a HEIDENHAIN 3-D touch probe (see the new Touch Probe Cycles Manual, chapter 2)
- Automatically by using a HEIDENHAIN 3-D touch probe (see the new Touch Probe Cycles Manual, chapter 3)

6 Start the part program in the operating mode Program Run, Full Sequence

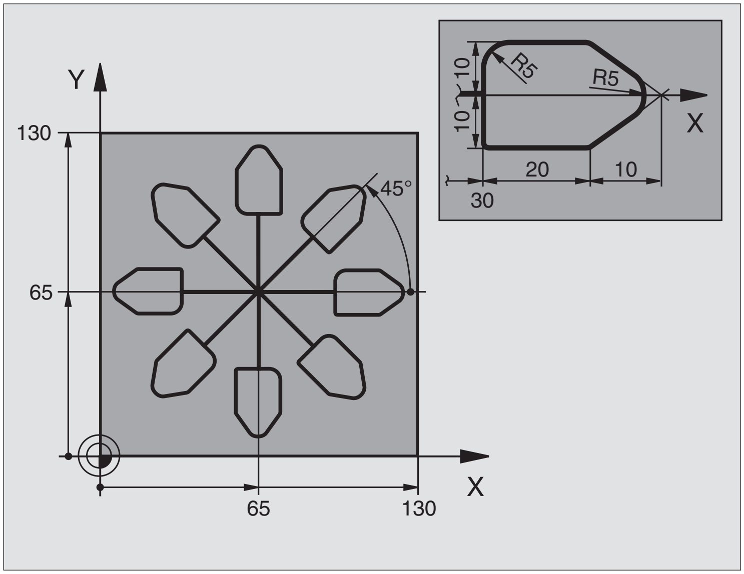
7 Manual Operation mode

Use the 3D-ROT soft key to set the function TILT WORKING PLANE to INACTIVE. Enter an angular value of 0° for each axis in the menu see "To activate manual tilting:," page 29.

Example: Coordinate transformation cycles

Program sequence

- Program the coordinate transformations in the main program
- For subprograms within a subprogram, see "Subprograms," page 317



%KOUMR G71 *	
N10 G30 G17 X+0 Y+0 Z-20 *	Define the workpiece blank
N20 G31 G90 X+130 Y+130 Z+0 *	
N30 G99 T1 L+0 R+1 *	Define the tool
N40 T1 G17 S4500 *	Tool call
N50 G00 G40 G90 Z+250 *	Retract the tool
N60 G54 X+65 Y+65 *	Shift datum to center
N70 L1.0 *	Call milling operation
N80 G98 L10 *	Set label for program section repeat
N90 G73 G91 H+45 *	Rotate by 45° (incremental)
N100 L1.0 *	Call milling operation
N110 L10.6 *	Return jump to LBL 10; execute the milling operation six times
N120 G73 G90 H+0 *	Reset the rotation
N130 G54 X+0 Y+0 *	Reset the datum shift
N140 G00 Z+250 M2 *	Retract in the tool axis, end program
N150 G98 L1 *	Subprogram 1:
N160 G00 G40 X+0 Y+0 *	Define milling operation
N170 Z+2 M3 *	
N180 G01 Z-5 F200 *	
N190 G41 X+30 *	

N200 G91 Y+10 *	
N210 G25 R5 *	
N220 X+20 *	
N230 X+10 Y-10 *	
N240 G25 R5 *	
N250 X-10 Y-10 *	
N260 X-20 *	
N270 Y+10 *	
N280 G40 G90 X+0 Y+0 *	
N290 G00 Z+20 *	
N300 G98 L0 *	
N999999 %KOUMR G71 *	

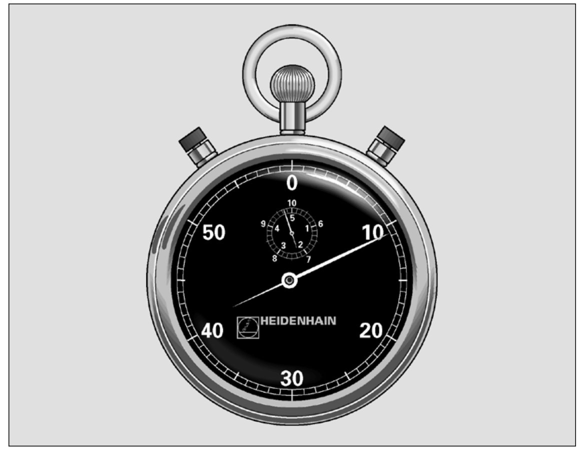
8.10 Special Cycles

DWELL TIME (Cycle G04)

This causes the execution of the next block within a running program to be delayed by the programmed dwell time. A dwell time can be used for such purposes as chip breaking.

Effect

Cycle 9 becomes effective as soon as it is defined in the program. Modal conditions such as spindle rotation are not affected.



- **`Dwell time in seconds:`** Enter the dwell time in seconds.

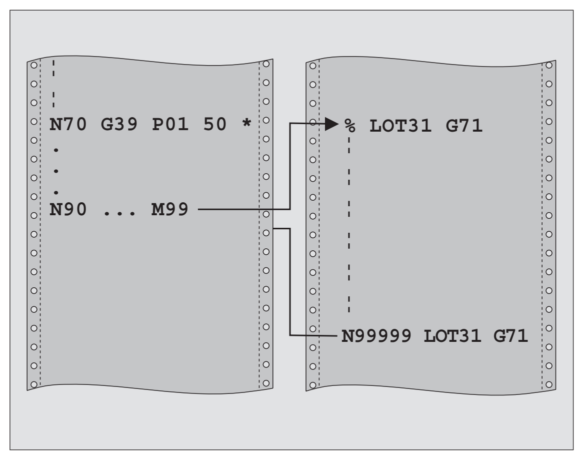
Input range 0 to 3600 s (1 hour) in 0.001 s steps

Example: NC block

```
N74 G04 F1.5 *
```

PROGRAM CALL (Cycle G39)

Routines that you have programmed (such as special drilling cycles or geometrical modules) can be written as main programs and then called like fixed cycles.



> **Before programming, note the following:**
>
> If you want to define an ISO program to be a cycle, enter the file type .I behind the program name.
>
> **Not TNC 410**
>
> If the program you are defining to be a cycle is located in the same directory as the program you are calling it from, you need only to enter the program name.
>
> If the program you are defining to be a cycle is not located in the same directory as the program you are calling it from, you must enter the complete path (for example TNC:\KLAR35\FK1\50.I.

- **`Program name:`** Enter the name of the program you want to call and, if necessary, the directory it is located in.

Call the program with

- **`G79`** (separate block) or
- **`M99`** (blockwise) or
- **`M89`** (executed after every positioning block)

Example: NC blocks

```
N550 G39 P01 50 *
N560 G00 X+20 Y+50 M9 9*
```

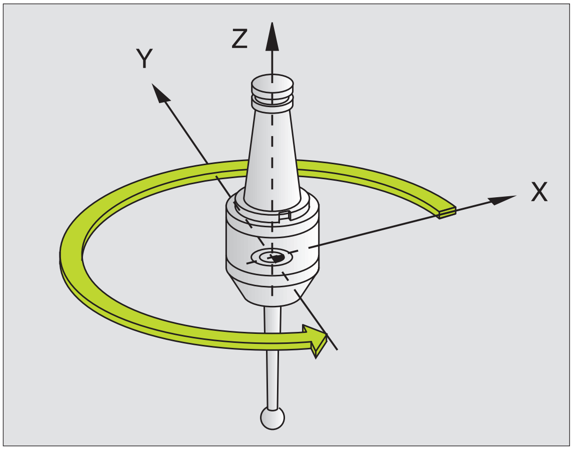
Example: Program call

A callable program 50 is to be called into a program via a cycle call.

ORIENTED SPINDLE STOP (Cycle G36)

Machine and control must be specially prepared by the machine tool builder for use of this cycle.

Cycle 13 is used internally for machining cycles 202, 204 and 209. Please note that, if required, you must program Cycle 13 again in your NC program after one of the machining cycles mentioned above.



The TNC can control the machine tool spindle and rotate it to a given angular position.

Oriented spindle stops are required for

- Tool changing systems with a defined tool change position
- Orientation of the transmitter/receiver window of HEIDENHAIN 3-D touch probes with infrared transmission

Effect

The angle of orientation defined in the cycle is positioned to by entering M19 or M20 (depending on the machine).

If you program M19 or M20 without having defined Cycle G36, the TNC positions the machine tool spindle to an angle that has been set in a machine parameter (see your machine manual).

- **Angle of orientation:** Enter the angle according to the reference axis of the working plane.

  Input range: 0 to 360°

  Input resolution: 0.001°

Example: NC block

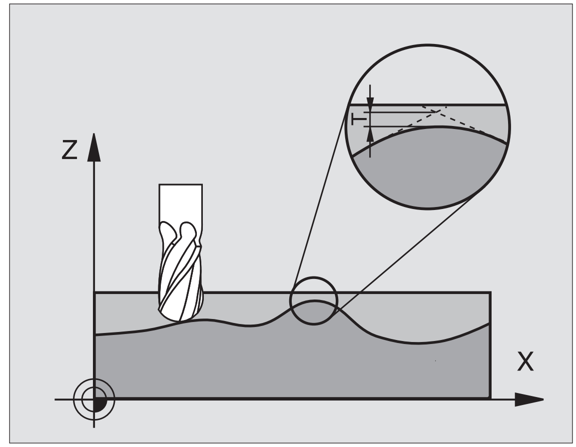
```
N76 G36 S25*
```

TOLERANCE (Cycle G62, not TNC 410)

Machine and control must be specially prepared by the machine tool builder for use of this cycle.

The TNC automatically smoothes the contour between two path elements (whether compensated or not). The tool has constant contact with the workpiece surface. If necessary, the TNC automatically reduces the programmed feed rate so that the program can be machined at the fastest possible speed without short pauses for computing time. As a result the surface quality is improved and the machine is protected.

A contour deviation results from the smoothing out. The size of this deviation **(tolerance value)** is set in a machine parameter by the machine manufacturer. You can change the pre-set tolerance value with Cycle G62.



Before programming, note the following:

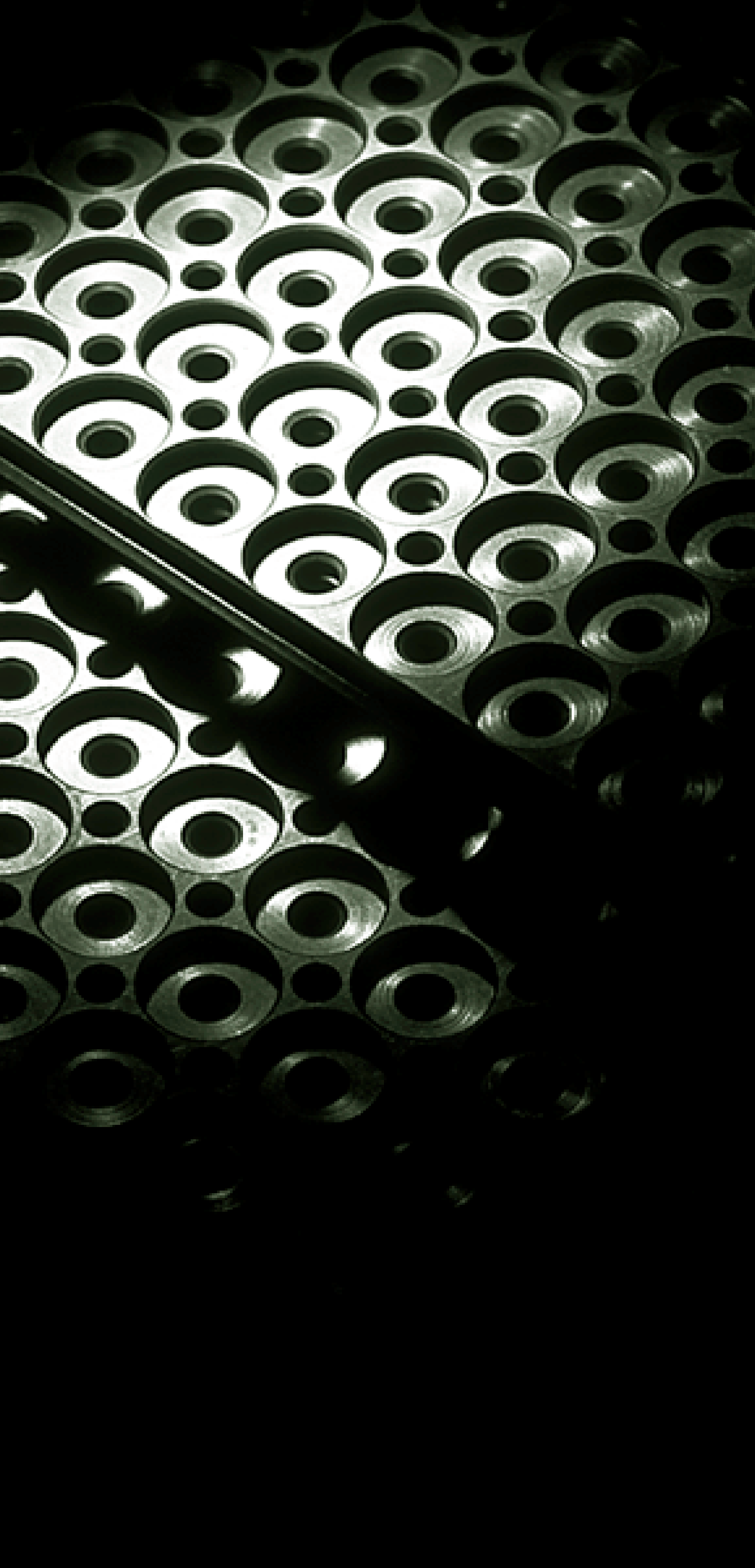
Cycle **G62** is DEF active which means that it becomes effective as soon as it is defined in the part program.

You can reset Cycle **G62** by defining it again and confirming the dialog question after the **tolerance value** with NO ENT.. Resetting Cycle 32 reactivates the pre-set tolerance:

- **Tolerance value**: Permissible contour deviation in mm

Example: NC block

```
N78 G62 T0.05*
```



9

Programming: Subprograms and Program Section Repeats

9.1 Labeling Subprograms and Program Section Repeats

Subprograms and program section repeats enable you to program a machining sequence once and then run it as often as desired.

Labels

The beginnings of subprograms and program section repeats are marked in a part program by G98 labels.

A label is identified by a number between 1 and 254. Each label can be set only once with G98 in a program.

If a label is set more than once, the TNC sends an error message at the end of the G98 block.

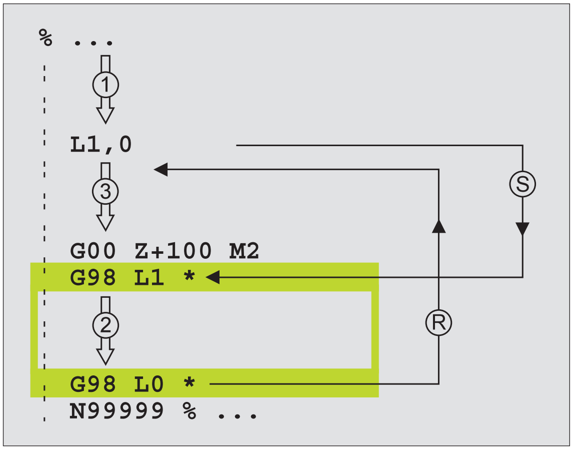
For the TNC 426, TNC 430:

With very long programs, you can limit the number of blocks to be checked for repeated labels with MP7229.

Label 0 (**G98 L0**) is used exclusively to mark the end of a subprogram and can therefore be used as often as desired.

9.2 Subprograms

Operating sequence



1 The TNC executes the part program up to the block in which a subprogram is called with **LN.0**. *n* can be any label number.
2 The subprogram is then executed from beginning to end. The subprogram end is marked **G98 L0**.
3 The TNC then resumes the part program from the block after the subprogram call **LN.0**.

Programming notes

- A main program can contain up to 254 subprograms.
- You can call subprograms in any sequence and as often as desired.
- A subprogram cannot call itself.
- Write subprograms at the end of the main program (behind the block with M2 or M30).
- If subprograms are located before the block with M02 or M30, they will be executed at least once even if they are not called.

Programming a subprogram

G 98

▶ To mark the beginning, select the function **G98** and confirm with the ENT key.

▶ Enter the subprogram number and confirm with the END key.

▶ To mark the end, select the function **G98** and enter the label number "0".

Calling a subprogram

L

▶ To call a subprogram, press the L key.

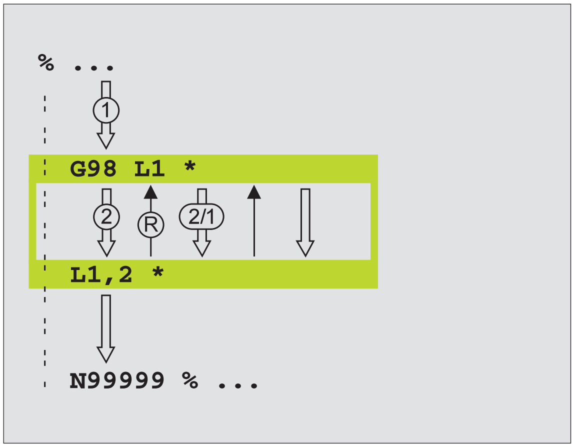
▶ Enter the label number for the subprogram you are calling and "**.0**".

> **L0.0** is not permitted, as it corresponds to the program end call.

9.3 Program Section Repeats

Label G98

The beginning of a program section repeat is marked by the label **G98 L**. A program section repeat ends with Ln,m, where m is the number of repeats.



Operating sequence

1. The TNC executes the part program up to the end of the program section (**L1.2**).
2. Then the program section between the called label and the label call **L 1.2** is repeated the number of times entered after the decimal point.
3. The TNC then resumes the part program after the last repetition.

Programming notes

- You can repeat a program section up to 65 534 times in succession.
- The TNC always executes the program section once more than the programmed number of repeats.

Programming a program section repeat

G 98

- To mark the beginning, select the function **G98** and confirm with the ENT key.
- Enter a label number for the program section to be repeated and confirm with the END key.

Calling a program section repeat

L

- Press the L key. Enter the label number for the program section to be repeated, and the number of repetitions after the comma.

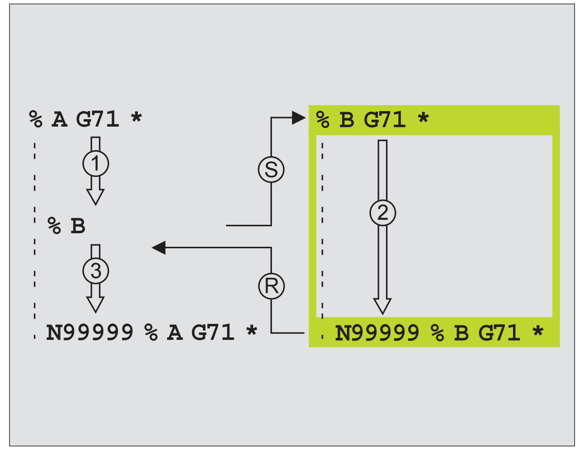
9.4 Separate Program as Subprogram

Operating sequence

1 The TNC executes the part program up to the block in which another program is called with %.

2 Then the other program is run from beginning to end.

3 The TNC then resumes the first (calling) part program with the block behind the program call.



Programming notes

- No labels are needed to call any program as a subprogram.
- The called program must not contain the miscellaneous functions M2 or M30.
- The called program must not contain a call with % into the calling program (endless loop).

Calling any program as a subprogram

%

▶ To select the program call functions, press the % key, enter the name of the program you wish to call and confirm your entry with the END key.

> You can also call a program with Cycle G39.
>
> If you want to call a conversational dialog program, enter the file type .H behind the program name.
>
> **For the TNC 426, TNC 430:**
>
> The program you are calling must be stored on the hard disk of your TNC.
>
> You need only enter the program name if the program you want to call is located in the same directory as the program you are calling it from.
>
> If the called program is not located in the same directory as the program you are calling it from, you must enter the complete path, e.g. TNC:\ZW35\ROUGH\PGM1.H

9.5 Nesting

Types of nesting

- Subprograms within a subprogram
- Program section repeats within a program section repeat
- Subprograms repeated
- Program section repeats within a subprogram

Nesting depth

The nesting depth is the number of successive levels in which program sections or subprograms can call further program sections or subprograms.

- Maximum nesting depth for subprograms: 8
- Maximum nesting depth for calling main programs: 4
- You can nest program section repeats as often as desired.

Subprogram within a subprogram

Example NC blocks

%UPGMS G71 *	
...	
N170 L1.0 *	Subprogram at label G98 L1 is called.
...	
N350 G00 G40 Z+100 M2 *	Last program block of the
	main program (with M2)
N360 G98 L1 *	Beginning of subprogram 1
...	
N390 L2.0 *	Subprogram at label G98 L2 is called.
...	
N450 G98 L0 *	End of subprogram 1
N460 G98 L2 *	Beginning of subprogram 2
...	
N620 G98 L0 *	End of subprogram 2
N999999 %UPGMS G71*	

Program execution

1. Main program UPGMS is executed up to block N170.
2. Subprogram 1 is called, and executed up to block N390.
3. Subprogram 2 is called, and executed up to block N620. End of subprogram 2 and return jump to the subprogram from which it was called.
4. Subprogram 1 is executed from block N400 up to block N450. End of subprogram 1 and return jump to the main program SUBPGMS.
5. Main program UPGMS is executed from block N180 up to block N350. Return jump to block 1 and end of program.

Repeating program section repeats

Example NC blocks

`%REPS G71 *`	
`...`	
`N150 G98 L1 *`	Beginning of program section repeat 1
`...`	
`N200 G98 L2 *`	Beginning of program section repeat 2
`...`	
`N270 L2.2 *`	The program section between this block and G98 L2
`...`	(block N200) is repeated twice.
`N350 L1.1 *`	The program section between this block and G98 L1
`...`	(block N150) is repeated once.
`N999999 %REPS G71 *`	

Program execution

1. Main program REPS is executed up to block N270.
2. Program section between block N270 and block N200 is repeated twice.
3. Main program REPS is executed from block N280 to block N350.
4. Program section between block N350 and block N150 is repeated once (including the program section repeat between block N200 and block N270).
5. Main program REPS is executed from block N360 to block N999999 (end of program).

Repeating a subprogram

Example NC blocks

%SUBPGREP G71 *	
...	
N100 G98 L1 *	Beginning of program section repeat 1
N110 L2.0 *	Subprogram call
N120 L1.2 *	The program section between this block and G98 L1
...	(block N100) is repeated twice.
N190 G00 G40 Z+100 M2*	Last block of the main program with M2
N200 G98 L2 *	Beginning of subprogram
...	
N280 G98 L0 *	End of subprogram
N999999 %SUBPGREP G71 *	

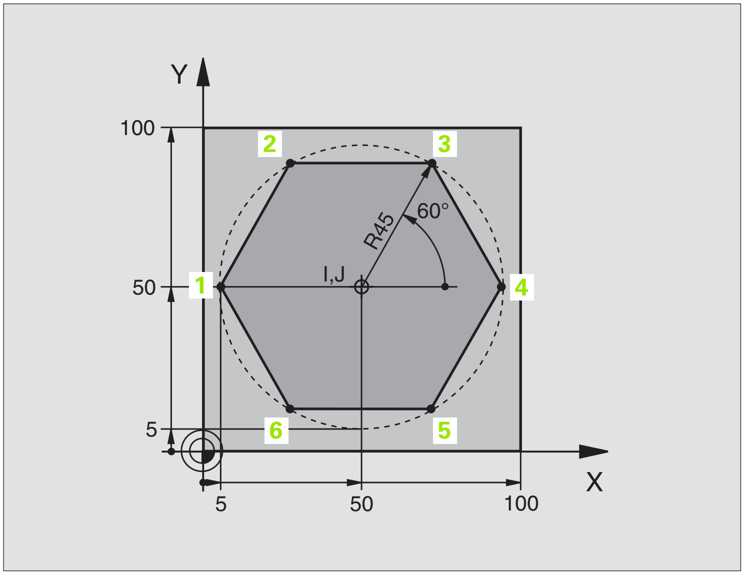
Program execution

1. Main program UPGREP is executed up to block N110.
2. Subprogram 2 is called and executed.
3. Program section between block N120 and block N100 is repeated twice. Subprogram 2 is repeated twice.
4. Main program SUBPGREP is executed once from block N130 to block N190. End of program.

Example: Milling a contour in several infeeds

Program sequence

- Pre-position the tool to the workpiece surface
- Enter the infeed depth in incremental values
- Mill the contour
- Repeat downfeed and contour-milling



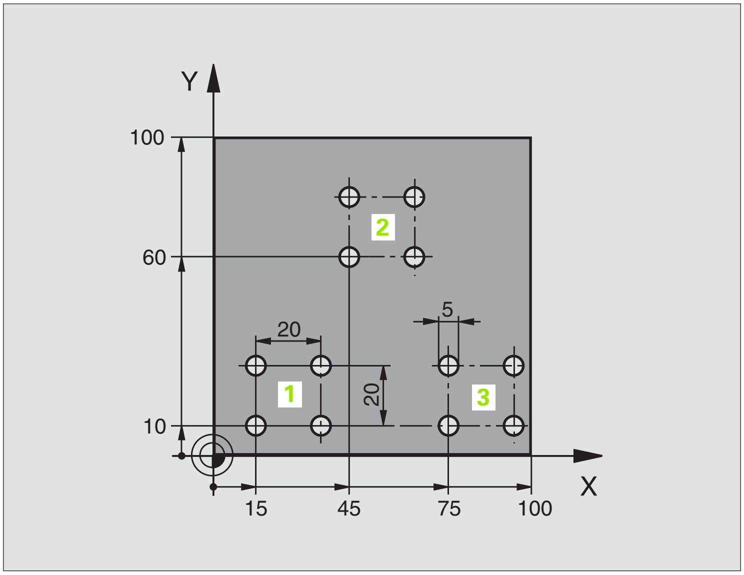
`%PGMWDH G71 *`	
`N10 G30 G17 X+0 Y+0 Z-40 *`	
`N20 G31 G90 X+100 Y+100 Z+0 *`	
`N30 G99 T1 L+0 R+7.5 *`	Define the tool
`N40 T1 G17 S4000 *`	Tool call
`N50 G00 G40 G90 Z+250 *`	Retract the tool
`N60 I+50 J+50 *`	Set pole
`N70 G10 R+60 H+180 *`	Pre-position in the working plane
`N80 G01 Z+0 F1000 M3 *`	Pre-position to the workpiece surface

`N90 G98 L1 *`	Set label for program section repeat
`N100 G91 Z-4 *`	Infeed depth in incremental values (in the open)
`N110 G11 G41 G90 R+45 H+180 F250 *`	First contour point
`N120 G26 R5 *`	Approach contour
`N130 H+120 *`	
`N140 H+60 *`	
`N150 H+0 *`	
`N160 H-60 *`	
`N170 H-120 *`	
`N180 H+180 *`	
`N190 G27 R5 F500 *`	Depart contour
`N200 G40 R+60 H+180 F1000 *`	Retract tool
`N210 L1.4 *`	Return jump to label 1; section is repeated a total of 4 times
`N220 G00 Z+250 M2 *`	Retract in the tool axis, end program
`N999999 %PGMWDH G71 *`	

Example: Groups of holes

Program sequence

- Approach the groups of holes in the main program
- Call the group of holes (subprogram 1)
- Program the group of holes only once in subprogram 1



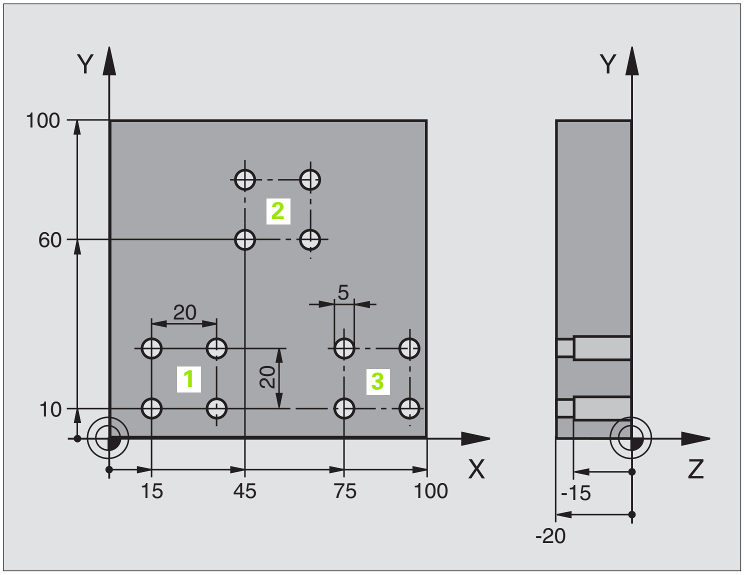
%UP1 G71 *	
N10 G30 G17 X+0 Y+0 Z-40 *	
N20 G31 G90 X+100 Y+100 Z+0 *	
N30 G99 T1 L+0 R+2.5 *	Define the tool
N40 T1 G17 S5000 *	Tool call
N50 G00 G40 G90 Z+250 *	Retract the tool
N60 G200	Cycle definition: drilling
Q200=2	set-up clearance
Q201=-30	Depth
Q206=300	Feed rate
Q202=5	Plunging depth
Q210=0	Dwell time at top
Q203=0	Workpiece surface
Q204=2	2nd set-up clearance
Q211=0 *	Dwell time at depth

N70 X+15 Y+10 M3 *	Move to starting point for group 1
N80 L1.0 *	Call the subprogram for the group
N90 X+45 Y+60 *	Move to starting point for group 2
N100 L1.0 *	Call the subprogram for the group
N110 X+75 Y+10 *	Move to starting point for group 3
N120 L1.0 *	Call the subprogram for the group
N130 G00 Z+250 M2 *	End of main program
N140 G98 L1 *	Beginning of subprogram 1: Group of holes
N150 G79 *	Call cycle for 1st hole
N160 G91 X+20 M99 *	Move to 2nd hole, call cycle
N170 Y+20 M99 *	Move to 3rd hole, call cycle
N180 X-20 G90 M99 *	Move to 4th hole, call cycle
N190 G98 L0 *	End of subprogram 1
N999999 %UP1 G71 *	

Example: Group of holes with several tools

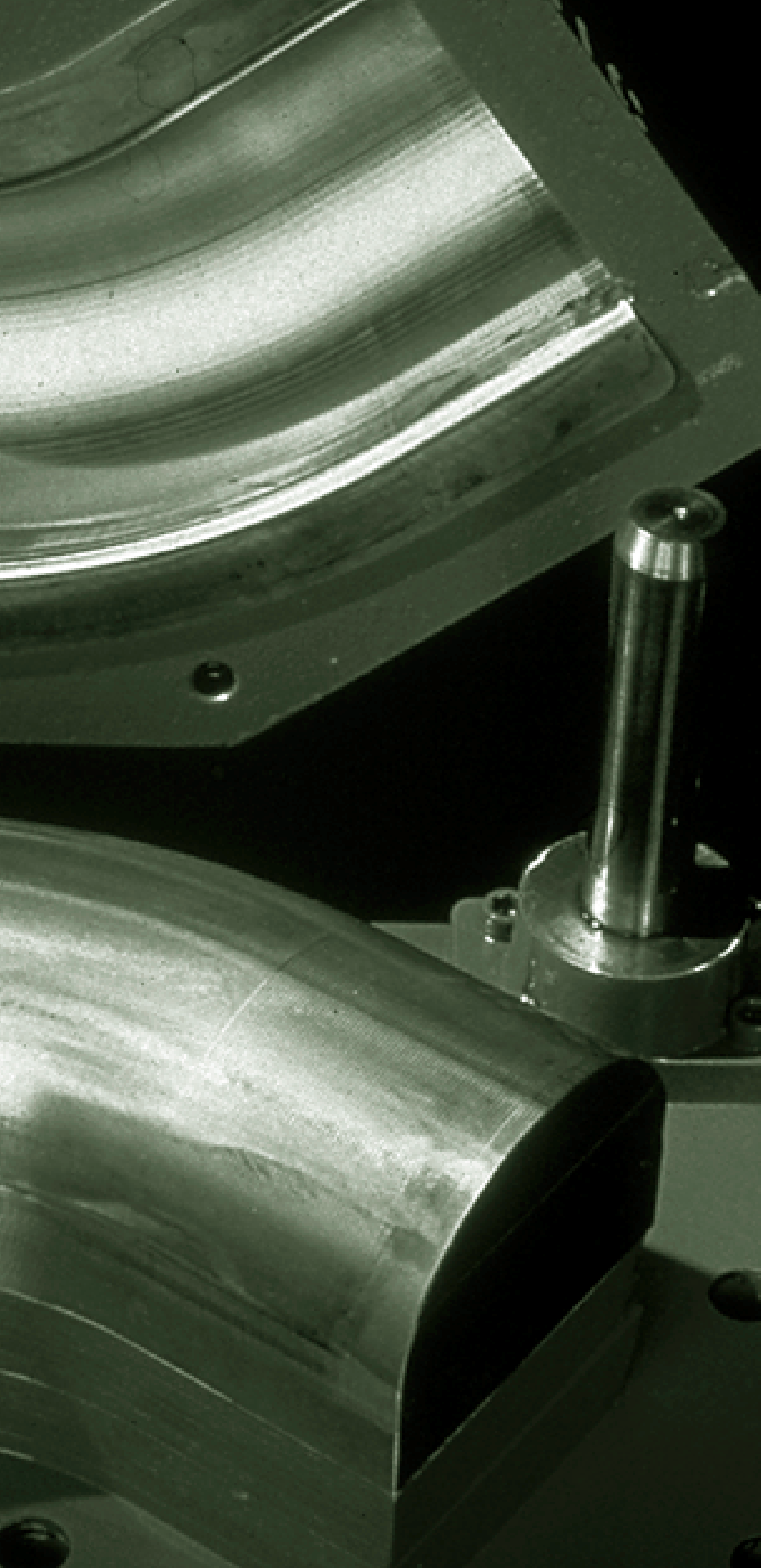
Program sequence

- Program the fixed cycles in the main program
- Call the entire hole pattern (subprogram 1)
- Approach the groups of holes in subprogram 1, call group of holes (subprogram 2)
- Program the group of holes only once in subprogram 2



%UP2 G71 *	
N10 G30 G17 X+0 Y+0 Z-40 *	
N20 G31 G90 X+100 Y+100 Z+0 *	
N30 G99 T1 L+0 R+4 *	Define tool: center drill
N40 G99 T2 L+0 R+3 *	Define tool: drill
N50 G99 T3 L+0 R+3.5 *	Define tool: reamer
N60 T1 G17 S5000 *	Call tool: center drill
N70 G00 G40 G90 Z+250 *	Retract the tool
N80 G200	Cycle definition: Centering
Q200=2	set-up clearance
Q201=-3	Depth
Q206=250	Feed rate
Q202=3	Plunging depth
Q210=0	Dwell time at top
Q203=+0	Coordinate of the workpiece surface
Q204=10	2nd set-up clearance
Q211=0.25	Dwell time at depth
N90 L1.0 *	Call subprogram 1 for the entire hole pattern

N100 G00 Z+250 M6 *	Tool change
N110 T2 G17 S4000 *	Call the drilling tool
N120 D0 Q201 P01 -25 *	New depth for drilling
N130 D0 Q202 P01 +5 *	New plunging depth for drilling
N140 L1.0 *	Call subprogram 1 for the entire hole pattern
N150 G00 Z+250 M6 *	Tool change
N160 T3 G17 S500 *	Tool call: reamer
N170 G201	Cycle definition: REAMING
Q200=2	set-up clearance
Q201=-15	Depth
Q206=250	Feed rate
Q211=0.5	Dwell time at depth
Q208=400	Retraction feed rate
Q203=+0	Coordinate of the workpiece surface
Q204=10 *	2nd set-up clearance
N180 L1.0 *	Call subprogram 1 for the entire hole pattern
N190 G00 Z+250 M2 *	End of main program
N200 G98 L1 *	Beginning of subprogram 1: Entire hole pattern
N210 G00 G40 G90 X+15 Y+10 M3 *	Move to starting point for group 1
N220 L2.0 *	Call subprogram 2 for the group
N230 X+45 Y+60 *	Move to starting point for group 2
N240 L2.0 *	Call subprogram 2 for the group
N250 X+75 Y+10 *	Move to starting point for group 3
N260 L2.0 *	Call subprogram 2 for the group
N270 G98 L0 *	End of subprogram 1
N280 G98 L2 *	Beginning of subprogram 2: Group of holes
N290 G79 *	Call cycle for 1st hole
N300 G91 X+20 M99 *	Move to 2nd hole, call cycle
N310 Y+20 M99 *	Move to 3rd hole, call cycle
N320 X-20 G90 M99 *	Move to 4th hole, call cycle
N330 G98 L0 *	End of subprogram 2
N340 END PGM UP2 MM	

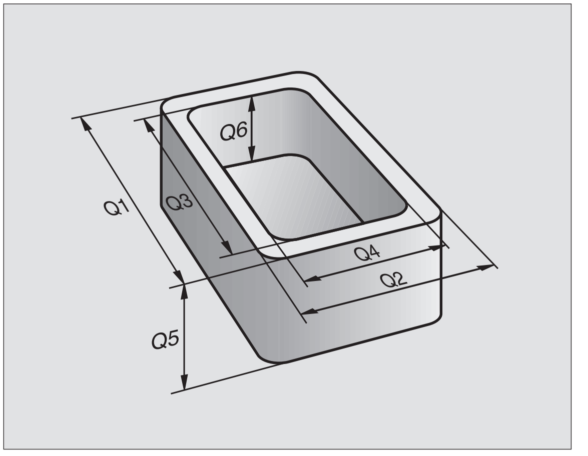


10

Programming: Q Parameters

10.1 Principle and Overview

You can program an entire family of parts in a single part program. You do this by entering variables called Q parameters instead of fixed numerical values.



Q parameters can represent information such as:

- Coordinate values
- Feed rates
- RPM
- Cycle data

Q parameters also enable you to program contours that are defined through mathematical functions. You can also use Q parameters to make the execution of machining steps depend on logical conditions.

Q parameters are designated by the letter Q and a number between 0 and 299. They are grouped according to three ranges:

Meaning	Range
Freely applicable parameters, globally effective for all programs stored in the TNC memory	Q0 to Q99
Parameters for special TNC functions	Q100 to Q199
Parameters that are primarily used for cycles, globally effective for all programs that are stored in the TNC memory	Q200 to Q399 (TNC 410: to Q299)

Programming notes

You can mix Q parameters and fixed numerical values within a program.

Q parameters can be assigned numerical values between -99 999.9999 and +99 999.9999. Internally, the TNC can calculate up to a width of 57 bits before and 7 bits after the decimal point (32-bit data width corresponds to a decimal value of 4 294 967 296).

Some Q parameters are always assigned the same data by the TNC. For example, Q108 is always assigned the current tool radius; see “Preassigned Q Parameters,” page 351.

If you are using the parameters Q60 to Q99 in OEM cycles, define via MP7251 whether the parameters are only to be used locally in the OEM cycles, or may be used globally.

Calling Q parameter functions

TNC 426, TNC 430: Press the PARAMETER soft key while you are entering a part program.

TNC 410: Press the "Q" key (to be found among the keys for value input and axis selection, beneath the –/+ key).

The TNC then displays the following soft keys:

Function group	Soft key
Basic arithmetic (assign, add, subtract, multiply, divide, square root)	BASIC ARITHM.
Trigonometric functions	TRIGO-NOMETRY
If/then conditions, jumps	JUMP
Other functions	DIVERSE FUNCTION
Entering formulas directly	FORMULA

10.2 Part Families—Q Parameters in Place of Numerical Values

The Q parameter function D0: ASSIGN assigns numerical values to Q parameters. This enables you to use variables in the program instead of fixed numerical values.

Example NC blocks

`N150 D00 Q10 P01 +25*`	Assign
`...`	Q10 contains the value 25
`N250  G00 X +Q10*`	corresponds to G00 X +25

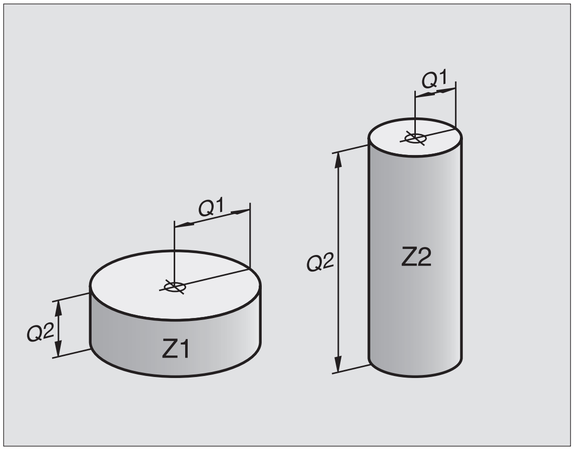
You need write only one program for a whole family of parts, entering the characteristic dimensions as Q parameters.

To program a particular part, you then assign the appropriate values to the individual Q parameters.

Example

Cylinder with Q parameters

Cylinder radius	R = Q1
Cylinder height	H = Q2
Cylinder Z1	Q1 = +30 Q2 = +10
Cylinder Z2	Q1 = +10 Q2 = +50



10.3 Describing Contours through Mathematical Operations

Function

The Q parameters listed below enable you to program basic mathematical functions in a part program:

- To select the Q parameter function, press the PARAMETER soft key on the TNC 426 / 430 or press the Q key on the TNC 410 (in the numerical keypad at right). The Q parameter functions are displayed in a soft-key row.
- To select the mathematical functions: Press the BASIC ARITHMETIC soft key. The TNC then displays the following soft keys:

Overview

Function	Soft key
D00: ASSIGN Example: **D00 Q5 P01 +60 *** Assigns a numerical value.	D0 X = Y
D01: ADDITION Example: **D01 Q1 P01 -Q2 P02 -5 *** Calculates and assigns the sum of two values.	D1 X + Y
D02: SUBTRACTION Example: **D02 Q1 P01 +10 P02 +5 *** Calculates and assigns the difference of two values.	D2 X - Y
D03: MULTIPLICATION Example: **D03 Q2 P01 +3 P02 +3 *** Calculates and assigns the product of two values.	D3 X * Y
D04: DIVISION Example: **D04 Q4 P01 +8 P02 +Q2 *** Calculates and assigns the quotient of two values. Not permitted: division by 0	D4 X / Y
D05: SQUARE ROOT Example: **D05 Q50 P01 4 *** Calculates and assigns the square root of a number. **Not permitted:** Square root of a negative number	D5 SQRT

To the right of the "=" character you can enter the following:

- Two numbers
- Two Q parameters
- A number and a Q parameter

The Q parameters and numerical values in the equations can be entered with positive or negative signs.

Programming fundamental operations

Programming example 1:

PARA-METER Q

To select Q parameter functions, press the PARAMETER soft key or the Q key..

BASIC ARITHM.

To select the mathematical functions: Press the BASIC ARITHMETIC soft key.

D0 X = Y

To select the Q parameter function ASSIGN, press the D0 X = Y soft key.

Parameter no. for result?

5 ENT

Enter the number of the Q parameter, e.g. 5.

1st value or parameter ?

10 ENT

Assign the value 10 to Q5.

Example: NC block

```
N16 D00 P01 +10 *
```

Programming example 2:

PARAMETER | Q — To select Q parameter functions, press the PARAMETER soft key or the Q key..

BASIC ARITHM. — To select the mathematical functions: Press the BASIC ARITHMETIC soft key.

D3 X * Y — To select the Q parameter function MULTIPLICATION, press the D03 X * Y soft key.

```
Parameter no. for result?
```

12 ENT — Enter the number of the Q parameter, e.g. 12.

```
1st value or parameter ?
```

Q5 ENT — Enter Q5 for the first value.

```
2nd value or parameter ?
```

7 ENT — Enter 7 for the second value.

Example: NC block

```
N17 D03 Q12 P01 +Q5 P02 +7 *
```

10.4 Trigonometric Functions

Definitions

Sine, cosine and tangent are terms designating the ratios of sides of right triangles. In this case:

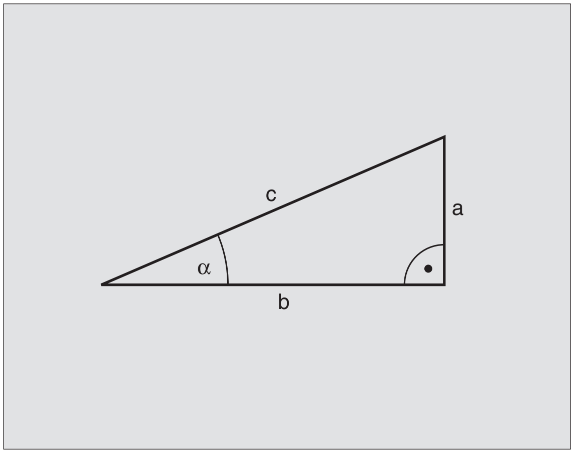
Sine: $\sin \alpha = a / c$
Cosine: $\cos \alpha = b / c$
Tangent: $\tan \alpha = a / b = \sin \alpha / \cos \alpha$

where

- c is the side opposite the right angle
- a is the side opposite the angle α
- b is the third side.

The TNC can find the angle from the tangent

$\alpha = \arctan \alpha = \arctan (a / b) = \arctan (\sin \alpha / \cos \alpha)$



Example:

a = 10 mm

b = 10 mm

$\alpha = \arctan (a / b) = \arctan 1 = 45°$

Furthermore:

$a^2 + b^2 = c^2$ (where $a^2 = a \times a$)

$c = \sqrt{(a^2 + b^2)}$

Programming trigonometric functions

Press the TRIGONOMETRY soft key to call the trigonometric functions. The TNC then displays the soft keys that are listed in the table below.

Programming: Compare "Example: Programming fundamental operations."

Function	Soft key
D06: SINE Example: **`D06 Q20 P01 -Q5 *`** Calculate the sine of an angle in degrees (°) and assign it to a parameter.	D6 SIN(X)
D07: COSINE Example: **`D07 Q21 P01 -Q5 *`** Calculate the cosine of an angle in degrees (°) and assign it to a parameter.	D7 COS(X)
D08: ROOT SUM OF SQUARES Example: **`D08 Q10 P01 +5 P02 +4 *`** Calculate and assign length from two values.	D8 X LEN Y
D13: ANGLE Example: **`D13 Q20 P01 +10 P02 -Q1 *`** Calculate the angle from the arc tangent of two sides or from the sine and cosine of the angle (0 < angle < 360°) and assign it to a parameter.	D13 X ANG Y

10.5 If-Then Decisions with Q Parameters

Function

The TNC can make logical If-Then decisions by comparing a Q parameter with another Q parameter or with a numerical value. If the condition is fulfilled, the TNC continues the program at the label that is programmed after the condition (for information on labels, see "Labeling Subprograms and Program Section Repeats," page 316). If it is not fulfilled, the TNC continues with the next block.

To call another program as a subprogram, enter a program call with % after label G98.

Unconditional jumps

An unconditional jump is programmed by entering a conditional jump whose condition is always true. Example:

D09 P01 +10 P02 +10 P03 1 *

Programming If-Then decisions

Press the JUMP soft key to call the If-Then conditions. The TNC then displays the following soft keys:

Function	Soft key
D09: IF EQUAL, JUMP Example: **`D09 P01 +Q1 P02 +Q3 P03 5 *`** If the two values or parameters are equal, jump to the given label.	D9 IF X EQ Y GOTO
D10: IF NOT EQUAL, JUMP Example: **`D10 P01 +10 P02 -Q5 P03 10 *`** If the two values or parameters are not equal, jump to the given label.	D10 IF X NE Y GOTO
D11: IF GREATER THAN, JUMP Example: **`D11 P01 +Q1 P02 +10 P03 5 *`** If the first parameter or value is greater than the second value or parameter, jump to the given label.	D11 IF X GT Y GOTO
D12: IF LESS THAN, JUMP Example: **`D12 P01 +Q5 P02 +0 P03 1 *`** If the first value or parameter is less than the second value or parameter, jump to the given label.	D12 IF X LT Y GOTO

Abbreviations used:

IF	:	If
EQU	:	Equals
NE	:	Not equal
GT	:	Greater than
LT	:	Less than
GOTO	:	Go to

10.6 Checking and Changing Q Parameters

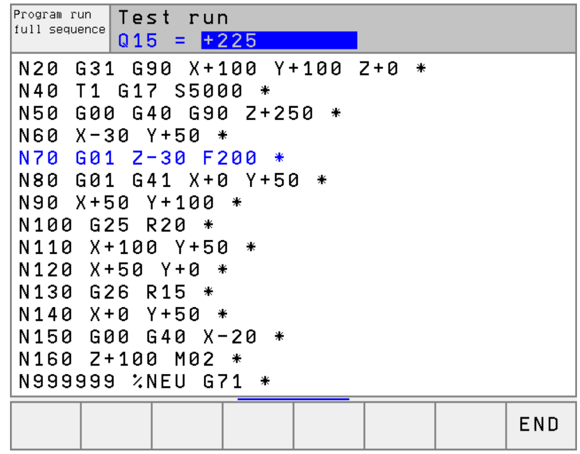
Procedure

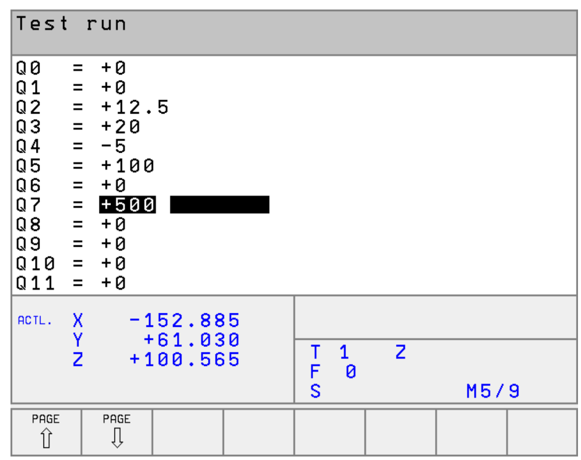
During a program run or test run, you can check or change Q parameters if necessary.

- If you are in a program run, interrupt it (for example, by pressing the machine STOPP button and the INTERNAL STOP soft key). If you are in a test run, interrupt it.

Q

- To call the Q parameter functions, press the Q key.
- TNC 426, TNC 430:
  Enter the Q parameter number and press the ENT key. The TNC displays the current value of the Q parameter in the dialog line.
- TNC 410:
  Using the arrow keys you can select a Q-parameter on the current screen page. With the PAGE soft keys, you can go to the next or the previous screen page.
- If you wish to change the value, enter a new value, confirm it with the ENT key and conclude your entry with the END key.
- To leave the value unchanged, terminate the dialog with the END key.





10.7 Additional Functions

Overview

Press the DIVERSE FUNCTION soft key to call the additional functions. The TNC then displays the following soft keys:

Function	Soft key
D14:ERROR Output error messages	D14 ERROR=
D15:PRINT Unformatted output of texts or Q parameter values	D15 PRINT
D19:PLC Transfer values to the PLC	D19 PLC=

D14: ERROR: Output error messages

Example NC block

The TNC is to display the text stored under error number 254.

```
N180 D14 P01 254 *
```

With the function D14: ERROR you can call messages under program control. The messages were preprogrammed by the machine tool builder or by HEIDENHAIN. If the TNC encounters a block with D 14 during program run, it will interrupt the run and display an error message. The program must then be restarted. The error numbers are listed in the table below.

Range of error numbers	Standard dialog text
0 ... 299	D 14: Error number 0 299
300 ... 999	Machine-dependent dialog
1000 ... 1099	Internal error messages (see table at right)

Error number	Text
1000	Spindle ?
1001	Tool axis is missing
1002	Slot width too large
1003	Tool radius too large
1004	Range exceeded
1005	Start position incorrect
1006	ROTATION not permitted

Error number	Text
1007	SCALING FACTOR not permitted
1008	MIRRORING not permitted
1009	Datum shift not permitted
1010	Feed rate is missing
1011	Entry value incorrect
1012	Wrong sign programmed
1013	Entered angle not permitted
1014	Touch point inaccessible
1015	Too many points
1016	Contradictory entry
1017	CYCL incomplete
1018	Plane wrongly defined
1019	Wrong axis programmed
1020	Wrong RPM
1021	Radius comp. undefined
1022	Rounding-off undefined
1023	Rounding radius too large
1024	Program start undefined
1025	Excessive subprogramming
1026	Angle reference missing
1027	No fixed cycle defined
1028	Slot width too small
1029	Pocket too small
1030	Q202 not defined
1031	Q205 not defined
1032	Enter Q218 greater than Q219
1033	CYCL 210 not permitted
1034	CYCL 211 not permitted
1035	Q220 too large

Error number	Text
1036	Enter Q222 greater than Q223
1037	Q244 must be greater than 0
1038	Q245 must not equal Q246
1039	Angle range must be < 360°
1040	Enter Q223 greater than Q222
1041	Q214: 0 not permitted
1042	Traverse direction not defined
1043	No datum table active
1044	Position error: center in axis 1
1045	Position error: center in axis 2
1046	Hole diameter too small
1047	Hole diameter too large
1048	Stud diameter too small
1049	Stud diameter too large
1050	Pocket too small: rework axis 1
1051	Pocket too small: rework axis 2
1052	Pocket too large: scrap axis 1
1053	Pocket too large: scrap axis 2
1054	Stud too small: scrap axis 1
1055	Stud too small: scrap axis 2
1056	Stud too large: rework axis 1
1057	Stud too large: rework axis 2
1058	TCHPROBE 425: length exceeds max
1059	TCHPROBE 425: length below min
1060	TCHPROBE 426: length exceeds max
1061	TCHPROBE 426: length below min
1062	TCHPROBE 430: diameter too large
1063	TCHPROBE 430: diameter too small
1064	No measuring axis defined

Error number	Text
1065	Tool breakage tolerance exceeded
1066	Enter Q247 unequal 0
1067	Enter Q247 greater than 5
1068	Datum table?
1069	Enter direction Q351 unequal 0
1070	Thread depth too large
1071	Missing calibration data
1072	Tolerance exceeded
1073	Block scan active
1074	ORIENTATION not permitted
1075	3-D ROT not permitted
1076	Activate 3-D ROT
1077	Enter depth as a negative value

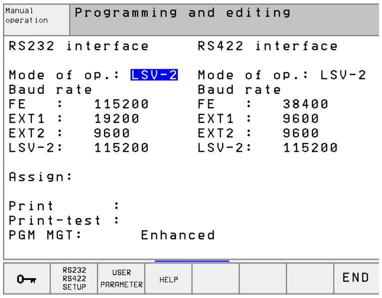
D15: PRINT: Output of texts or Q parameter values

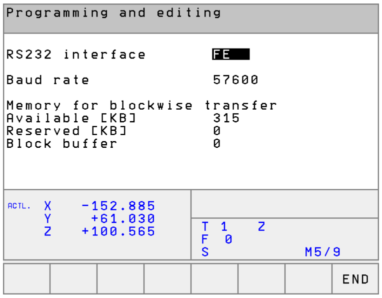
TNC 410:

In the menu option RS 232 interface, you must enter where the texts are to be stored, see “Setting the Data Interface for the TNC 410,” page 393.

TNC 426, TNC 430:

Setting the data interface: In the menu option PRINT or PRINT-TEST, you must enter the path for storing the texts or Q parameters, see “Assign,” page 396.





The function D15: PRINT transfers Q parameter values and error messages through the data interface, for example to a printer. When you save the data in the TNC memory or transfer them to a PC, the TNC stores the data in the file %FN 15RUN.A (output in program run mode) or in the file %FN15SIM.A (output in test run mode). The data are transmitted from a buffer. Data output begins at the latest by program end or when you stop the program. In the Single Block mode of operation, data transfer begins at block end.

Output dialog texts and error messages with D15: PRINT ”numerical value”

Numerical values from 0 to 99:	Dialog texts for OEM cycles
Numerical values exceeding 100:	PLC Error Messages

Example: Output of dialog text 20

```
N67 D15 P01 20 *
```

Output dialog texts and error messages with D15: PRINT ”Q parameter”

Application example: Recording workpiece measurement.

You can transfer up to six Q parameters and numerical values simultaneously.

Example: Output of dialog text 1 and numerical value for Q1

```
N70 D15 P01 1 P02 Q1 *
```

D19: PLC: Transferring values to the PLC

The function D19: PLC transfers up to two numerical values or Q parameter contents to the PLC.

Increments and units: 0.1 µm or 0.0001°

Example: Transfer the numerical value 10 (which means 1 µm or 0.001°) to the PLC

```
N56 D19 P01 +10 P02 +Q3 *
```

10.8 Entering Formulas Directly

Entering formulas

You can enter mathematical formulas that include several operations directly into the part program by soft key.

Press the FORMULA soft key to call the formula functions. The TNC displays the following soft keys in several soft-key rows:

Logic command	Soft key
Addition Example: **Q10 = Q1 + Q5**	+
Subtraction Example: **Q25 = Q7 – Q108**	–
Multiplication Example: **Q12 = 5 * Q5**	*
Division Example: **Q25 = Q1 / Q2**	/
Opening parenthesis Example: **Q12 = Q1 * (Q2 + Q3)**	(
Closing parenthesis Example: **Q12 = Q1 * (Q2 + Q3)**	)
Square of a value Example: **Q15 = SQ 5**	SQ
Square root Example: **Q22 = SQRT 25**	SQRT
Sine of an angle Example: **Q44 = SIN 45**	SIN
Cosine of an angle Example: **Q45 = COS 45**	COS
Tangent of an angle Example: **Q46 = TAN 45**	TAN
Arc sine Inverse of the sine. Determine the angle from the ratio of the opposite side to the hypotenuse. Example: **Q10 = ASIN 0.75**	ASIN
Arc cosine Inverse of the cosine. Determine the angle from the ratio of the adjacent side to the hypotenuse. Example: **Q11 = ACOS Q40**	ACOS

Logic command	Soft key
Arc tangent Inverse of the tangent. Determine the angle from the ratio of the opposite to the adjacent side. Example: **Q12 = ATAN Q50**	ATAN
Powers of values Example: **Q15 = 3^3**	^
Constant "pi" (3.14159) Example: **Q15 = PI**	PI
Natural logarithm (LN) of a number Base 2.7183 Example: **Q15 = LN Q11**	LN
Logarithm of a number, base 10 Example: **Q33 = LOG Q22**	LOG
Exponential function, 2.7183 to the power of n Example: **Q1 = EXP Q12**	EXP
Negate (multiplication by -1) Example: **Q2 = NEG Q1**	NEG
Truncate decimal places (form an integer) Example: **Q3 = INT Q42**	INT
Absolute value of a number Example: **Q4 = ABS Q22**	ABS
Truncate places before the decimal point (form a fraction) Example: **Q5 = FRAC Q23**	FRAC
Check algebraic sign of a number (not TNC 426, TNC 430) Example: **Q12 = SGN Q50** If result for Q12= 1: Q50 >= 0 If result for Q12= 0: Q50 < 0	SGN

Rules for formulas

Mathematical formulas are programmed according to the following rules:

Higher-level operations are performed first (multiplication and division before addition and subtraction)

```
N112  Q1 = 5 * 3 + 2 * 10 = 35
```

1st step 5 * 3 = 15
2nd step 2 * 10 = 20
3rd step 15 +20 = 35

or

```
N113  Q2 = SQ 10 - 3^3 = 73
```

1st step 10 squared = 100
2nd step 3 to the power of 3 = 27
3rd step 100 – 27 = 73

Distributive law

for calculating with parentheses

a * (b + c) = a * b + a * c

Programming example

Calculate an angle with the arc tangent from the opposite side (Q12) and adjacent side (Q13); then store in Q25.

Q PARA-METER To select Q parameter functions, press the Q key, or press the soft key PARAMETER.

FORMULA For formula input, press the FORMULA soft key.

Parameter no. for result?

ENT 25 Enter the parameter number.

▷ ATAN Shift the soft-key row and select the arc tangent function.

◁ (Shift the soft-key row and open the parentheses.

Q 12 Enter Q parameter number 12.

/ Select division.

Q 13 Enter Q parameter number 13.

) END Close parentheses and conclude formula entry.

Example NC block

```
N37  Q25 = ATAN (Q12/Q13)
```

10.9 Preassigned Q Parameters

The Q parameters Q100 to Q122 are assigned values by the TNC. These values include:

- Values from the PLC
- Tool and spindle data
- Data on operating status, etc.

Values from the PLC: Q100 to Q107

The TNC uses the parameters Q100 to Q107 to transfer values from the PLC to an NC program.

Active tool radius: Q108

The active value of the tool radius is assigned to Q108. Q108 is calculated from:

- Tool radius R (tool table or G99 block)
- Delta value DR from the tool table
- Delta value DR from the TOOL CALL block

Tool axis: Q109

The value of Q109 depends on the current tool axis:

Tool axis	Parameter value
No tool axis defined	Q109 = -1
X axis	Q109 = 0
Y axis	Q109 = 1
Z axis	Q109 = 2
U axis	Q109 = 6
V axis	Q109 = 7
W axis	Q109 = 8

Spindle status: Q110

The value of Q110 depends on which M function was last programmed for the spindle:

M Function	Parameter value
No spindle status defined	Q110 = -1
M03: Spindle ON, clockwise	Q110 = 0

M Function	Parameter value
M04: Spindle ON, counterclockwise	Q110 = 1
M05 after M03	Q110 = 2
M05 after M04	Q110 = 3

Coolant on/off: Q111

M Function	Parameter value
M08: Coolant ON	Q111 = 1
M09: Coolant OFF	Q111 = 0

Overlap factor: Q112

The overlap factor for pocket milling (MP7430) is assigned to Q112.

Unit of measurement for dimensions in the program: Q113

The value of parameter Q113 specifies whether the highest-level NC program (for nesting with %...) is programmed in millimeters or inches.

Dimensions of the main program	Parameter value
Metric system (mm)	Q113 = 0
Inch system (inches)	Q113 = 1

Tool length: Q114

The current value for the tool length is assigned to Q114.

Coordinates after probing during program run

The parameters Q115 to Q119 contain the coordinates of the spindle position at the moment of contact during programmed measurement with the 3-D touch probe. The coordinates are referenced to the datum that is currently active in the Manual operating mode.

The length and radius of the probe tip are not compensated in these coordinates.

Coordinate axis	Parameter value
X axis	Q115
Y axis	Q116
Z axis	Q117

Coordinate axis	Parameter value
IVth axis dependent on MP100	Q118
Vth axis (not TNC 410) dependent on MP100	Q119

Deviation between actual value and nominal value during automatic tool measurement with the TT 130

Actual-nominal deviation	Parameter value
Tool length	Q115
Tool radius	Q116

Tilting the working plane with mathematical angles (not TNC 410): Rotary axis coordinates calculated by the TNC

coordinates	Parameter value
A axis	Q120
B axis	Q121
C axis	Q122

Results of measurements with touch probe cycles

(see also Touch Probe Cycles User's Manual)

Measured actual values	Parameter value
Angle of a straight line	Q150
Center in reference axis	Q151
Center in minor axis	Q152
Diameter	Q153
Length of pocket	Q154
Width of pocket	Q155
Length in the axis selected in the cycle	Q156
Position of the center line	Q157
Angle of the A axis	Q158
Angle of the B axis	Q159
Coordinate of the axis selected in the cycle	Q160

Measured deviation	Parameter value
Center in reference axis	Q161
Center in minor axis	Q162
Diameter	Q163
Length of pocket	Q164
Width of pocket	Q165
Measured length	Q166
Position of the center line	Q167

Measured solid angle	Parameter value
Rotation about the A axis	Q170
Rotation about the B axis	Q171
Rotation about the C axis	Q172

Workpiece status	Parameter value
Good	Q180
Re-work	Q181
Scrap	Q182

Measured deviation with cycle 440	Parameter value
X axis	Q185
Y axis	Q186
Z axis	Q187

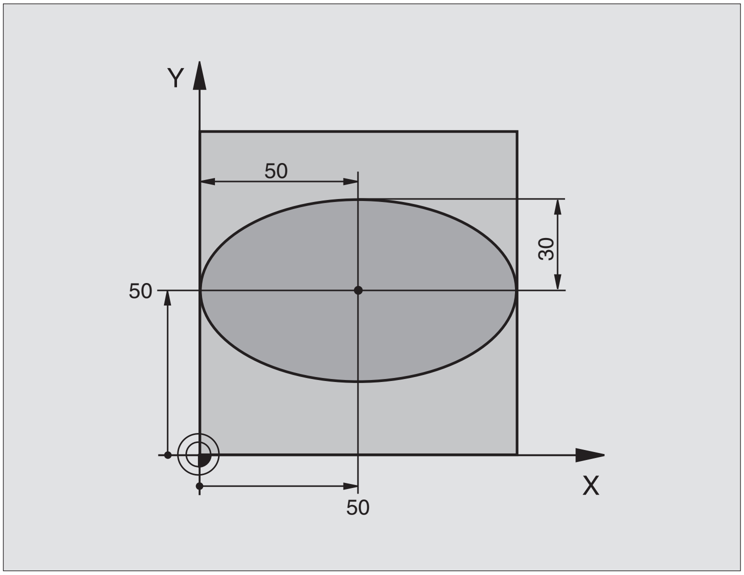
Reserved for internal use	Parameter value
Markers for cycles (point patterns)	Q197

Status during tool measurement with TT	Parameter value
Tool within tolerance	Q199 = 0.0
Tool is worn (LTOL/RTOL exceeded)	Q199 = 1.0
Tool is broken (LBREAK/RBREAK exceeded)	Q199 = 2.0

Example: Ellipse

Program sequence

- The contour of the ellipse is approximated by many short lines (defined in Q7). The more calculating steps you define for the lines, the smoother the curve becomes.
- The machining direction can be altered by changing the entries for the starting and end angles in the plane:
  Clockwise machining direction:
  starting angle > end angle
  Counterclockwise machining direction: starting angle < end angle
- The tool radius is not taken into account.



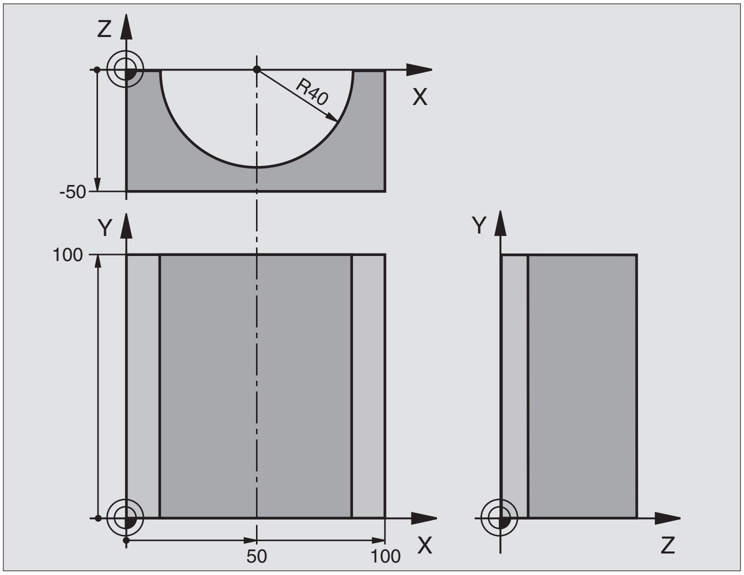
%ELLIPSIS G71 *	
N10 D00 Q1 P01 +50 *	Center in X axis
N20 D00 Q2 P01 +50 *	Center in Y axis
N30 D00 Q3 P01 +50 *	Semiaxis in X
N40 D00 Q4 P01 +30 *	Semiaxis in Y
N50 D00 Q5 P01 +0 *	Starting angle in the plane
N60 D00 Q6 P01 +360 *	End angle in the plane
N70 D00 Q7 P01 +40 *	Number of calculating steps
N80 D00 Q8 P01 +30 *	Rotational position of the ellipse
N90 D00 Q9 P01 +5 *	Milling depth
N100 D00 Q10 P01 +100 *	Feed rate for plunging
N110 D00 Q11 P01 +350 *	Feed rate for milling
N120 D00 Q12 P01 +2 *	Set-up clearance for pre-positioning
N130 G30 G17 X+0 Y+0 Z-20 *	Define the workpiece blank
N140 G31 G90 X+100 Y+100 Z+0 *	
N150 G99 T1 L+0 R+2.5 *	Define the tool
N160 T1 G17 S4000 *	Tool call
N170 G00 G40 G90 Z+250 *	Retract the tool
N180 L10.0 *	Call machining operation
N190 G00 Z+250 M2 *	Retract in the tool axis, end program

`N200 G98 L10 *`	Subprogram 10: Machining operation
`N210 G54 X+Q1 Y+Q2 *`	Shift datum to center of ellipse
`N220 G73 G90 H+Q8 *`	Account for rotational position in the plane
`N230 Q35 = (Q6 - Q5) / Q7`	Calculate angle increment
`N240 D00 Q36 P01 +Q5 *`	Copy starting angle
`N250 D00 Q37 P01 +0 *`	Set counter
`N260 Q21 = Q3 * COS Q36`	Calculate X coordinate for starting point
`N270 Q22 = Q4 * SIN Q36`	Calculate Y coordinate for starting point
`N280 G00 G40 X+Q21 Y+Q22 M3 *`	Move to starting point in the plane
`N290 Z+Q12 *`	Pre-position in tool axis to setup clearance
`N300 G01 Z-Q9 FQ10 *`	Move to working depth
`N310 G98 L1 *`	
`N320 Q36 = Q36 + Q35`	Update the angle
`N330 Q37 = Q37 + 1`	Update the counter
`N340 Q21 = Q3 * COS Q36`	Calculate the current X coordinate
`N350 Q22 = Q4 * SIN Q36`	Calculate the current Y coordinate
`N360 G01 X+Q21 Y+Q22 FQ11 *`	Move to next point
`N370 D12 P01 +Q37 P02 +Q7 P03 1 *`	Unfinished? If not finished return to label 1
`N380 G73 G90 H+0 *`	Reset the rotation
`N390 G54 X+0 Y+0 *`	Reset the datum shift
`N400 G00 G40 Z+Q12 *`	Move to setup clearance
`N410 G98 L0 *`	End of subprogram
`N999999 %ELLIPSIS G71 *`	

Example: Concave cylinder machined with spherical cutter

Program sequence

- Program functions only with a spherical cutter. The tool length refers to the sphere center.
- The contour of the cylinder is approximated by many short line segments (defined in Q13). The more line segments you define, the smoother the curve becomes.
- The cylinder is milled in longitudinal cuts (here: parallel to the Y axis).
- The machining direction can be altered by changing the entries for the starting and end angles in space:
  Clockwise machining direction:
  starting angle > end angle
  Counterclockwise machining direction: starting angle < end angle
- The tool radius is compensated automatically.



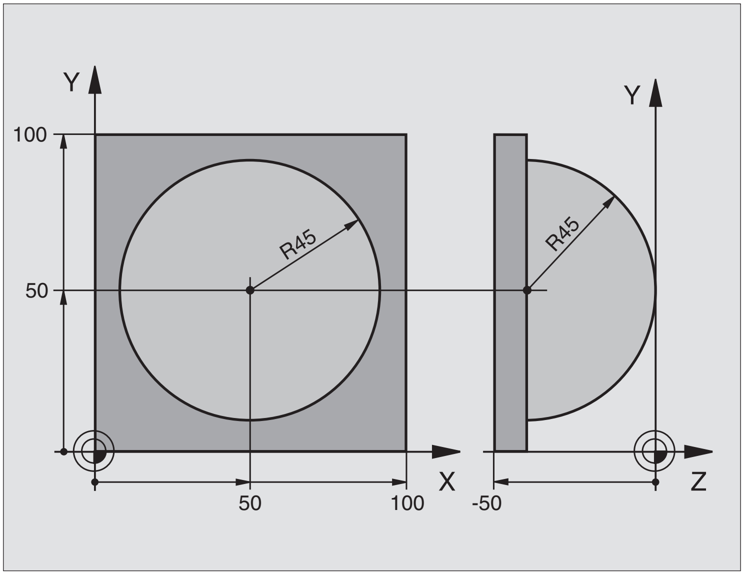
%CYLIN G71 *	
N10 D00 Q1 P01 +50 *	Center in X axis
N20 D00 Q2 P01 +0 *	Center in Y axis
N30 D00 Q3 P01 +0 *	Center in Z axis
N40 D00 Q4 P01 +90 *	Starting angle in space (Z/X plane)
N50 D00 Q5 P01 +270 *	End angle in space (Z/X plane)
N60 D00 Q6 P01 +40 *	Radius of the cylinder
N70 D00 Q7 P01 +100 *	Length of the cylinder
N80 D00 Q8 P01 +0 *	Rotational position in the X/Y plane
N90 D00 Q10 P01 +5 *	Allowance for cylinder radius
N100 D00 Q11 P01 +250 *	Feed rate for plunging
N110 D00 Q12 P01 +400 *	Feed rate for milling
N120 D00 Q13 P01 +90 *	Number of cuts
N130 G30 G17 X+0 Y+0 Z-50 *	Define the workpiece blank
N140 G31 G90 X+100 Y+100 Z+0 *	
N150 G99 T1 L+0 R+3 *	Define the tool
N160 T1 G17 S4000 *	Tool call
N170 G00 G40 G90 Z+250 *	Retract the tool
N180 L10.0 *	Call machining operation
N190 D00 Q10 P01 +0 *	Reset allowance

`N200 L10.0 *`	Call machining operation
`N210 G00 G40 Z+250 M2 *`	Retract in the tool axis, end program
`N220 G98 L10 *`	Subprogram 10: Machining operation
`N230 Q16 = Q6 - Q10 - Q108`	Account for allowance and tool, based on the cylinder radius
`N240 D00 Q20 P01 +1 *`	Set counter
`N250 D00 Q24 P01 +Q4 *`	Copy starting angle in space (Z/X plane)
`N260 Q25 = (Q5 - Q4) / Q13`	Calculate angle increment
`N270 G54 X+Q1 Y+Q2 Z+Q3 *`	Shift datum to center of cylinder (X axis)
`N280 G73 G90 H+Q8 *`	Account for rotational position in the plane
`N290 G00 G40 X+0 Y+0 *`	Pre-position in the plane to the cylinder center
`N300 G01 Z+5 F1000 M3 *`	Pre-position in the tool axis
`N310 G98 L1 *`	
`N320 I+0 K+0 *`	Set pole in the Z/X plane
`N330 G11 R+Q16 H+Q24 FQ11 *`	Move to starting position on cylinder, plunge-cutting obliquely into the material
`N340 G01 G40 Y+Q7 FQ12 *`	Longitudinal cut in Y+ direction
`N350 D01 Q20 P01 +Q20 P02 +1 *`	Update the counter
`N360 D01 Q24 P01 +Q24 P02 +Q25 *`	Update solid angle
`N370 D11 P01 +Q20 P02 +Q13 P03 99 *`	Finished? If finished, jump to end
`N380 G11 R+Q16 H+Q24 FQ11 *`	Move in an approximated "arc" for the next longitudinal cut
`N390 G01 G40 Y+0 FQ12 *`	Longitudinal cut in Y– direction
`N400 D01 Q20 P01 +Q20 P02 +1 *`	Update the counter
`N410 D01 Q24 P01 +Q24 P02 +Q25 *`	Update solid angle
`N420 D12 P01 +Q20 P02 +Q13 P03 1 *`	Unfinished? If not finished, return to LBL 1
`N430 G98 L99 *`	
`N440 G73 G90 H+0 *`	Reset the rotation
`N450 G54 X+0 Y+0 Z+0 *`	Reset the datum shift
`N460 G98 L0 *`	End of subprogram
`N999999 %CYLIN G71 *`	

Example: Convex sphere machined with end mill

Program sequence

- This program requires an end mill.
- The contour of the sphere is approximated by many short lines (in the Z/X plane, defined in Q14). The smaller you define the angle increment, the smoother the curve becomes.
- You can determine the number of contour cuts through the angle increment in the plane (defined in Q18).
- The tool moves upward in three-dimensional cuts.
- The tool radius is compensated automatically.



`%BALL G71 *`	
`N10 D00 Q1 P01 +50 *`	Center in X axis
`N20 D00 Q2 P01 +50 *`	Center in Y axis
`N30 D00 Q4 P01 +90 *`	Starting angle in space (Z/X plane)
`N40 D00 Q5 P01 +0 *`	End angle in space (Z/X plane)
`N50 D00 Q14 P01 +5 *`	Angle increment in space
`N60 D00 Q6 P01 +45 *`	Radius of the sphere
`N70 D00 Q8 P01 +0 *`	Starting angle of rotational position in the X/Y plane
`N80 D00 Q9 P01 +360 *`	End angle of rotational position in the X/Y plane
`N90 D00 Q18 P01 +10 *`	Angle increment in the X/Y plane for roughing
`N100 D00 Q10 P01 +5 *`	Allowance in sphere radius for roughing
`N110 D00 Q11 P01 +2 *`	Setup clearance for pre-positioning in the tool axis
`N120 D00 Q12 P01 +350 *`	Feed rate for milling
`N130 G30 G17 X+0 Y+0 Z-50 *`	Define the workpiece blank
`N140 G31 G90 X+100 Y+100 Z+0 *`	
`N150 G99 T1 L+0 R+7.5 *`	Define the tool
`N160 T1 G17 S4000 *`	Tool call
`N170 G00 G40 G90 Z+250 *`	Retract the tool
`N180 L10.0 *`	Call machining operation
`N190 D00 Q10 P01 +0 *`	Reset allowance

N200 D00 Q18 P01 +5 *	Angle increment in the X/Y plane for finishing
N210 L10.0 *	Call machining operation
N220 G00 G40 Z+250 M2 *	Retract in the tool axis, end program
N230 G98 L10 *	Subprogram 10: Machining operation
N240 D01 Q23 P01 +Q11 P02 +Q6 *	Calculate Z coordinate for pre-positioning
N250 D00 Q24 P01 +Q4 *	Copy starting angle in space (Z/X plane)
N260 D01 Q26 P01 +Q6 P02 +Q108 *	Compensate sphere radius for pre-positioning
N270 D00 Q28 P01 +Q8 *	Copy rotational position in the plane
N280 D01 Q16 P01 +Q6 P02 -Q10 *	Account for allowance in the sphere radius
N290 G54 X+Q1 Y+Q2 Z-Q16 *	Shift datum to center of sphere
N300 G73 G90 H+Q8 *	Account for starting angle of rotational position in the plane
N310 G98 L1 *	Pre-position in the tool axis
N320 I+0 J+0 *	Set pole in the X/Y plane for pre-positioning
N330 G11 G40 R+Q26 H+Q8 FQ12 *	Pre-position in the plane
N340 I+Q108 K+0 *	Set pole in the Z/X plane, offset by the tool radius
N350 G01 Y+0 Z+0 FQ12 *	Move to working depth
N360 G98 L2 *	
N370 G11 G40 R+Q6 H+Q24 FQ12 *	Move upward in an approximated “arc”
N380 D02 Q24 P01 +Q24 P02 +Q14 *	Update solid angle
N390 D11 P01 +Q24 P02 +Q5 P03 2 *	Inquire whether an arc is finished. If not finished, return to LBL 2.
N400 G11 R+Q6 H+Q5 FQ12 *	Move to the end angle in space
N410 G01 G40 Z+Q23 F1000 *	Retract in the tool axis
N420 G00 G40 X+Q26 *	Pre-position for next arc
N430 D01 Q28 P01 +Q28 P02 +Q18 *	Update rotational position in the plane
N440 D00 Q24 P01 +Q4 *	Reset solid angle
N450 G73 G90 H+Q28 *	Activate new rotational position
N460 D12 P01 +Q28 P02 +Q9 P03 1 *	Unfinished? If not finished, return to label 1
N470 D09 P01 +Q28 P02 +Q9 P03 1 *	
N480 G73 G90 H+0 *	Reset the rotation
N490 G54 X+0 Y+0 Z+0 *	Reset the datum shift
N500 G98 L0 *	End of subprogram
N999999 %BALL G71 *	

11

Test Run and Program Run

11.1 Graphics

Function

In the Program Run modes of operation as well as in the Test Run mode, the TNC provides the following display modes. Using soft keys, select whether you desire:

- Plan view
- Projection in 3 planes
- 3-D view

The TNC graphic depicts the workpiece as if it were being machined with a cylindrical end mill. If a tool table is active, you can also simulate the machining operation with a spherical cutter (not TNC 410). For this purpose, enter R2 = R in the tool table.

The TNC will not show a graphic if

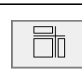
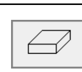
- the current program has no valid blank form definition
- no program is selected

With the TNC 426, TNC 430 you can use machine parameters 7315 to 7317 to have the TNC display a graphic even if no tool axis is defined or moved.

> A graphic simulation is not possible for program sections or programs in which rotary axis movements or a tilted working plane are defined. In this case, the TNC will display an error message.
>
> The TNC graphic does not show a radius oversize **DR** that has been programmed in the **T** block.
>
> The TNC can display the graphic only if the ratio of the shortest to the longest side of the blank form is less than 1 : 64.

Overview of display modes

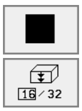
The TNC displays the following soft keys in the Program Run (not TNC 410) and Test Run modes of operation:

Display mode	Soft key
Plan view	
Projection in 3 planes	
3-D view	

Limitations during program run

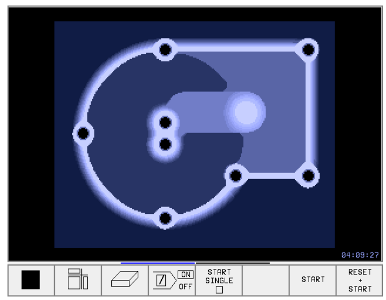
A graphical representation of a running program is not possible if the microprocessor of the TNC is already occupied with complicated machining tasks or if large areas are being machined. Example: Multipass milling over the entire blank form with a large tool. The TNC interrupts the graphics and displays the text **ERROR** in the graphics window. The machining process is continued, however.

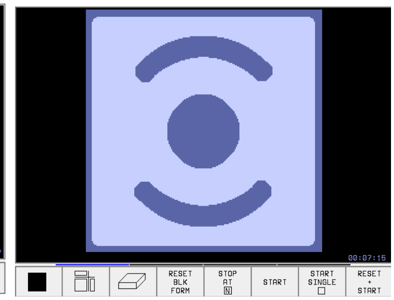
Plan view



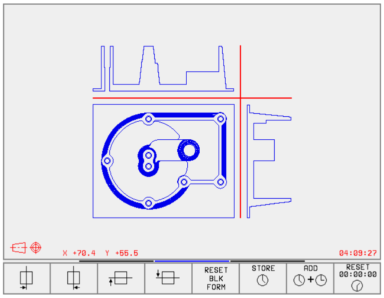
- Press the soft key for plan view.
- Select the number of depth levels (after shifting the soft-key row, not TNC 410). You can choose between 16 or 32 shades of depth. The deeper the surface, the darker the shade.

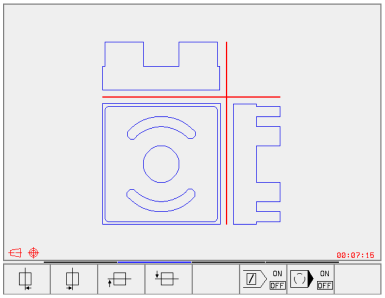
  This is the fastest of the three graphic display modes.





Projection in 3 planes



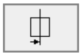
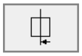
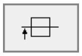
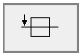


Similar to a workpiece drawing, the part is displayed with a plan view and two sectional planes. A symbol to the lower left indicates whether the display is in first angle or third angle projection according to ISO 6433 (selected with MP7310).

Details can be isolated in this display mode for magnification (not TNC 410, see "Magnifying details," page 367.

In addition, you can shift the sectional planes with the corresponding soft keys:

- Press the soft key for projection in three planes.
- Shift the soft-key row until the TNC displays the following soft keys:

Function	Soft keys	
Shift the vertical sectional plane to the right or left		
Shift the horizontal sectional plane upwards or downwards		

The positions of the sectional planes are visible during shifting.

Coordinates of the line of intersection (not TNC 410)

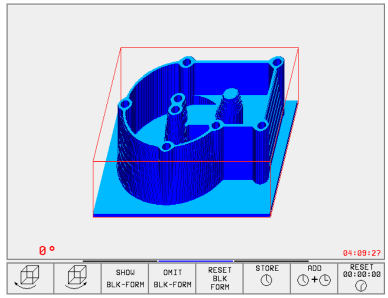
At the bottom of the graphics window, the TNC displays the coordinates of the line of intersection, referenced to the workpiece datum. Only the coordinates of the working plane are shown. This function is activated with Machine Parameter 7310.

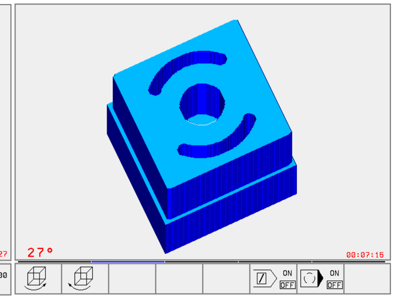
3-D view

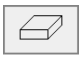
The workpiece is displayed in three dimensions, and can be rotated about the vertical axis.

In 3-D view, the workpiece can be rotated about the vertical axis. The shape of the workpiece blank can be depicted by a frame overlay at the beginning of the graphic simulation (not TNC 410).

In the Test Run mode of operation you can isolate details for magnification, see "Magnifying details," page 367.



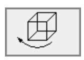




- Press the soft key for 3-D view.

To rotate the 3-D view

Shift the soft-key row until the following soft keys appear:

Function	Soft keys
Rotate the workpiece in 27° steps about the vertical axis	 

Switch the frame overlay display for the workpiece blank on/off (not TNC 410):

SHOW BLK-FORM

- Show the frame overlay with SHOW BLK FORM.

OMIT BLK-FORM

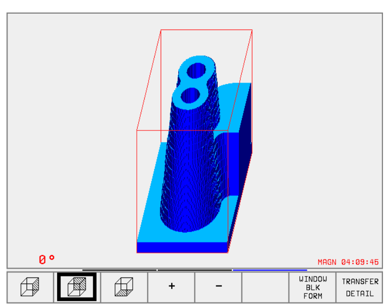
- Omit the frame overlay with OMIT BLK-FORM

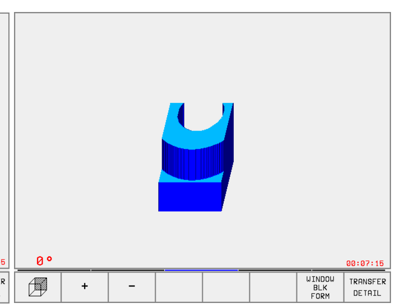
Magnifying details

You can magnify details in the Test Run mode of operation in the following display modes, provided that the graphic simulation is stopped:

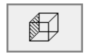
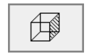
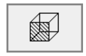
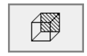
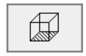
- Projection in three planes
- 3-D view

The graphic simulation must first have been stopped. A detail magnification is always effective in all display modes.





Shift the soft-key row in the Test Run mode of operation until the following soft keys appear:

Function	Soft keys	
Select the left/right workpiece surface		
Select the front/back workpiece surface		
Select the top/bottom workpiece surface		
Shift the sectional plane to reduce or magnify the blank form	–	+
Select the isolated detail	TRANSFER DETAIL	

Changing the detail magnification

The soft keys are listed in the table above.

- Interrupt the graphic simulation, if necessary.
- Select the workpiece surface with the corresponding soft key (see table).
- To reduce or magnify the blank form, press and hold the MINUS or PLUS soft key, respectively.
- Restart the test run or program run by pressing the START soft key (RESET + START returns the workpiece blank to its original state).

Cursor position during detail magnification (not TNC 410)

During detail magnification, the TNC displays the coordinates of the axis that is currently being isolated. The coordinates describe the area determined for magnification. To the left of the slash is the smallest coordinate of the detail (MIN point), to the left is the largest (MAX point).

If a graphic display is magnified, this is indicated with **MAGN** at the lower right of the graphics window.

If the workpiece blank cannot be further enlarged or reduced, the TNC displays an error message in the graphics window. To clear the error message, reduce or enlarge the workpiece blank.

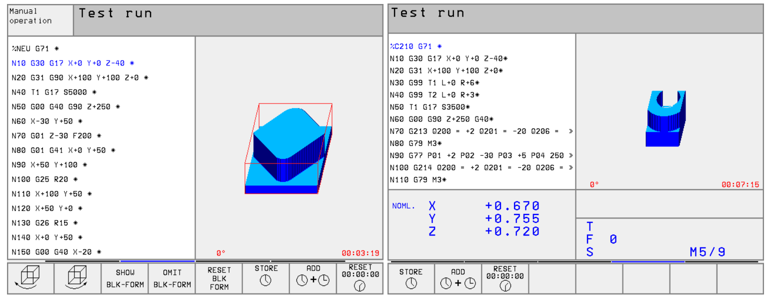
Repeating graphic simulation

A part program can be graphically simulated as often as desired, either with the complete workpiece or with a detail of it.

Function	Soft key
Restore workpiece blank to the detail magnification in which it was last shown.	RESET BLK FORM
Reset detail magnification so that the machined workpiece or workpiece blank is displayed as the blank form was programmed.	WINDOW BLK FORM

☞ With the WINDOW BLK FORM soft key, you return the displayed workpiece blank to its originally programmed dimensions, even after isolating a detail—without TRANSFER DETAIL.

Measuring the machining time



Program Run modes of operation

The timer counts and displays the time from program start to program end. The timer stops whenever machining is interrupted.

Test Run

The timer displays the approximate time which the TNC calculates from the duration of tool movements. The time calculated by the TNC cannot be used for calculating the production time because the TNC does not account for the duration of machine-dependent interruptions, such as tool change.

Activating the stopwatch function

Shift the soft-key rows until the TNC displays the following soft keys with the stopwatch functions:

Stopwatch functions	Soft key
Store displayed time	STORE
Display the sum of stored time and displayed time	ADD
Clear displayed time	RESET 00:00:00

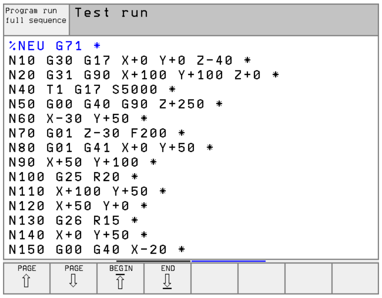
The soft keys available to the left of the stopwatch functions depend on the selected screen layout.

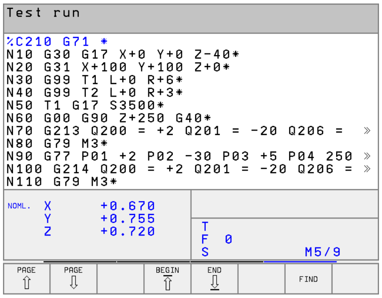
11.2 Functions for Program Display

Overview

In the program run modes of operation as well as in the Test Run mode, the TNC provides the following soft keys for displaying a part program in pages:

Function	Soft key
Go back in the program by one screen	PAGE ⇧
Go forward in the program by one screen	PAGE ⇩
Go to beginning of program	BEGIN ⇧
Go to end of program	END ⇩





11.3 Test Run

Function

In the Test Run mode of operation you can simulate programs and program sections to prevent errors from occurring during program run. The TNC checks the programs for the following:

- Geometrical incompatibilities
- Missing data
- Impossible jumps
- Violation of the machine's working space

The following functions are also available:

- Blockwise test run
- Interrupt test at any block
- Optional block skip
- Functions for graphic simulation
- Measuring the machining time
- Additional status display

Running a program test

If the central tool file is active, a tool table must be active (status S) to run a program test. Select a tool table via the file manager (PGM MGT) in the Test Run mode of operation.

With the MOD function BLANK IN WORK SPACE, you can activate work space monitoring for the test run, see “Showing the Workpiece in the Working Space (not TNC 410),” page 408.

- ▶ Select the Test Run mode of operation.
- ▶ Call the file manager with the PGM MGT key and select the file you wish to test, or
- ▶ Go to the program beginning: Select line “0” with the GOTO key and confirm your entry with the ENT key.

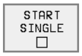
The TNC then displays the following soft keys:

Function	Soft key
Test the entire program	START
Test each program block individually	START SINGLE
Show the blank form and test the entire program	RESET + START
Interrupt the test run	STOP

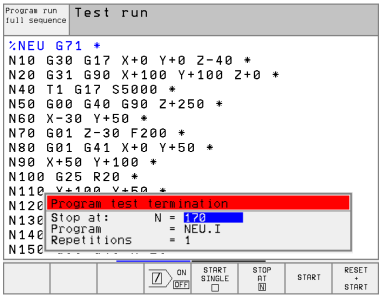
Run a program test up to a certain block

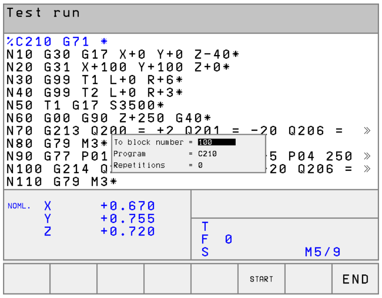
With the STOP AT N function the TNC does a test run up to the block with block number N.

- Go to the beginning of program in the Test Run mode of operation.
- To run a program test up to a specific block, press the STOP AT N soft key.



- **Stop at N:** Enter the block number at which you wish the test to stop.
- **Program:** Enter the name of the program that contains the block with the selected block number. The TNC displays the name of the selected program. If the test run is to be interrupted in a program that was called with %, you must enter this name.
- **Repetitions:** If N is located in a program section repeat, enter the number of repeats that you want to run.
- To test a program section, press the START soft key. The TNC will test the program up to the entered block.





11.4 Program Run

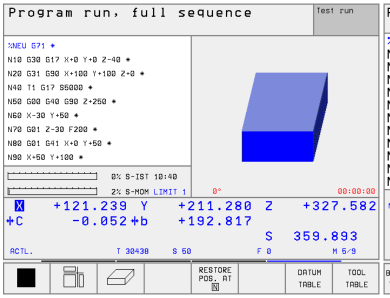
Function

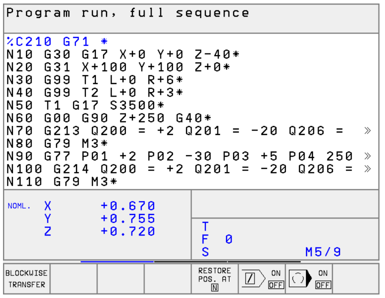
In the Program Run, Full Sequence mode the TNC executes a part program continuously to its end or up to a program stop.

In the Program Run, Single Block mode of operation you must start each block separately by pressing the machine START button.

The following TNC functions can be used in the program run modes of operation:

- Interrupt program run
- Start program run from a certain block
- Optional block skip
- Editing the tool table TOOL.T
- Checking and changing Q parameters
- Superimposing handwheel positioning (not TNC 410)
- Functions for graphic simulation (not TNC 410)
- Additional status display





Running a part program

Preparation

1 Clamp the workpiece to the machine table.

2 Set the datum.

3 Select the necessary tables and pallet files (status M).

4 Select the part program (status M).

> You can adjust the feed rate and spindle speed with the override knobs.
>
> **For the TNC 426, TNC 430:**
>
> It is possible to reduce the rapid traverse speed when starting the NC program using the FMAX soft key. The entered value remains in effect even after the machine has been turned off and on again. In order to re-establish the original rapid traverse speed, you need to re-enter the corresponding value.

Program Run, Full Sequence

- Start the part program with the machine START button.

Program Run, Single Block

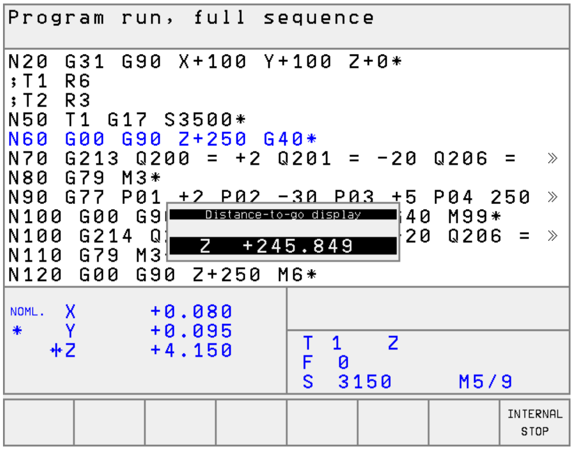
- Start each block of the part program individually with the machine START button.

Running a part program containing coordinates from non-controlled axes (not TNC 426, TNC 430)

Function

The TNC can also run programs in which you have programmed non-controlled axes.

If the TNC arrives at a block in which you have programmed a non-controlled axis, it stops program run. At the same time it superimposes a window showing the distance-to-go to the target position (see figure at top right).



Procedure

When the TNC displays the distance-to-go window, proceed as follows:

- Move the axis manually to the target position. The TNC constantly updates the distance-to-go window, and always shows the distance remaining to reach the target position.
- Once you have reached the target position, press the NC START key to continue program run. If you press the NC START key before you have arrived at the target position, the TNC will output an error message.

Machine parameter 1030.x determines how accurately you need to approach the target position (possible input values: 0.001 to 2 mm).

Non-controlled axes must be programmed in separate positioning blocks, otherwise the TNC will output an error message.

Interrupting machining

There are several ways to interrupt a program run:

- Programmed interruptions
- Machine STOP button
- Switching to Program Run, Single Block

If the TNC registers an error during program run, it automatically interrupts the machining process.

Programmed interruptions

You can program interruptions directly in the part program. The TNC interrupts the program run at a block containing one of the following entries:

- G38
- Miscellaneous function M0, M2 or M30
- Miscellaneous function M6 (determined by the machine tool builder)

Interrupting the machining process with the machine STOP button

- Press the machine STOP button: The block which the TNC is currently executing is not completed. The asterisk in the status display blinks.
- If you do not wish to continue the machining process, you can reset the TNC with the INTERNAL STOP soft key. The asterisk in the status display goes out. In this case, the program must be restarted from the program beginning.

Interrupting the machining process by switching to the Program Run, Single Block mode of operation.

You can interrupt a program that is being run in the Program Run, Full Sequence mode of operation by switching to Program Run, Single Block. The TNC interrupts the machining process at the end of the current block.

Moving the machine axes during an interruption

You can move the machine axes during an interruption in the same way as in the Manual Operation mode.

TNC 426, TNC 430: Caution - Risk of collision!

If you interrupt program run while the working plane is tilted, you can change from a tilted to a non-tilted coordinate system, and vice versa, by pressing the 3-D ON/OFF soft key.

The functions of the axis direction buttons, the electronic handwheel and the positioning logic for return to contour are then evaluated by the TNC. When retracting the tool make sure the correct coordinate system is active and the angular values of the tilt axes are entered in the 3-D ROT menu.

Application example:
Retracting the spindle after tool breakage

- Interrupt machining.
- Enable the external direction keys: Press the MANUAL OPERATION soft key.
- Move the axes with the machine axis direction buttons.

For the TNC 426, TNC 430:

On some machines you may have to press the machine START button after the MANUAL OPERATION soft key to enable the axis direction buttons. Refer to your machine manual.

Resuming program run after an interruption

> If a program run is interrupted during a fixed cycle, the program must be resumed from the beginning of the cycle. This means that some machining operations will be repeated.
>
> If you interrupt a program run during execution of a subprogram or program section repeat, use the RESTORE POS AT N function to return to the position at which the program run was interrupted.

When a program run is interrupted, the TNC stores:

- The data of the last defined tool
- Active coordinate transformations (e.g. datum shift, rotation, mirroring)
- The coordinates of the circle center that was last defined

> Note that the stored data remains active until it is reset (e.g. if you select a new program).

The stored data are used for returning the tool to the contour after manual machine axis positioning during an interruption (RESTORE POSITION soft key).

Resuming program run with the START button

You can resume program run by pressing the machine START button if the program was interrupted in one of the following ways:

- The machine STOP button was pressed.
- An interruption was programmed.

Resuming program run after an error

If the error message is not blinking:

- ▶ Remove the cause of the error.
- ▶ To clear the error message from the screen, press the CE key.
- ▶ Restart the program, or resume program run where it was interrupted.

If the error message is blinking:

- ▶ Press and hold the END key for two seconds. This induces a TNC system restart.
- ▶ Remove the cause of the error.
- ▶ Start again.

If you cannot correct the error, write down the error message and contact your repair service agency.

Mid-program startup (block scan)

The RESTORE POS AT N feature must be enabled and adapted by the machine tool builder. Refer to your machine manual.

With the RESTORE POS AT N feature (block scan) you can start a part program at any block you desire. The TNC scans the program blocks up to that point. Machining can be graphically simulated.

If you have interrupted a part program with an INTERNAL STOP, the TNC automatically offers the interrupted block N for mid-program startup.

Mid-program startup must not begin in a subprogram.

All necessary programs, tables and pallet files must be selected in a program run mode of operation (status M).

If the program contains a programmed interruption before the startup block, the block scan is interrupted. Press the machine START button to continue the block scan.

After a block scan, return the tool to the calculated position with RESTORE POSITION.

Tool length compensation does not take effect until after the tool call and a following positioning block; this also applies to an altered tool length.

For the TNC 426, TNC 430:

If you are working with nested programs, you can use Machine Parameter 7680 to define whether the block scan is to begin at block 0 of the main program or at block 0 of the last interrupted program.

The function M128 is not permitted during a mid-program startup.

If the working plane is tilted, you can use the 3-D ON/OFF soft key to define whether the TNC is to return to the contour in a tilted or in a non-tilted coordinate system.

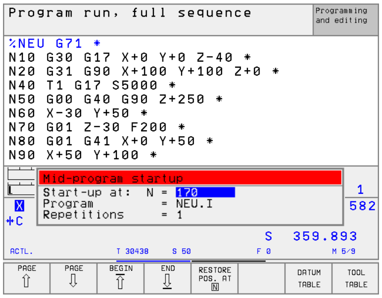
If you want to use the block scan feature in a pallet table, select the program in which a mid-program startup is to be performed from the pallet table by using the arrow keys. Then press the RESTORE POS AT N soft key.

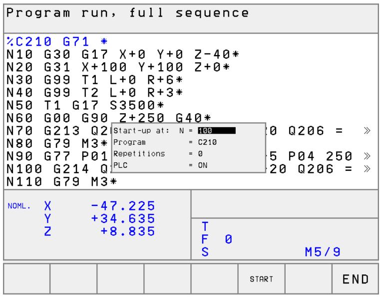
All touch probe cycles and the Cycle 247 are skipped in a mid-program startup. Result parameters that are written to from these cycles might therefore remain empty.

- To go to the first block of the current program to start a block scan, enter GOTO "0".
- To select mid-program startup, press the RESTORE POS AT N soft key.

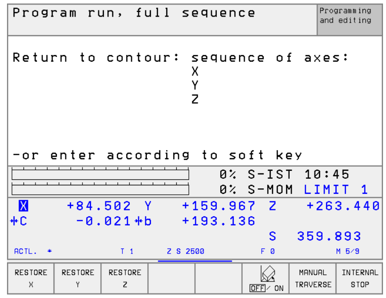
RESTORE POS. AT N

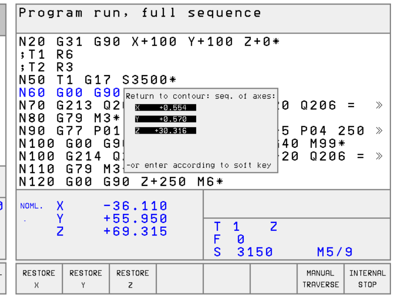
- **Start-up at N:** Enter the block number N at which the block scan should end.
- **Program:** Enter the name of the program containing block N.
- **Repetitions:** If block N is located in a program section repeat, enter the number of repetitions to be calculated in the block scan.
- PLC ON/OFF (not TNC 426, TNC 430): To account for tool calls and miscellaneous functions M: Set the PLC to ON (use the ENT key to switch between ON and OFF). If PLC is set to OFF, the TNC considers only the geometry. The tool in the spindle must equal the tool called by the program.
- To start the block scan,
  TNC 426, TNC 430: Press the machine START key.
  TNC 410: Press the START soft key.
- To return to the contour, see “Returning to the contour,” page 382





Returning to the contour





With the RESTORE POSITION function, the TNC returns to the workpiece contour in the following situations:

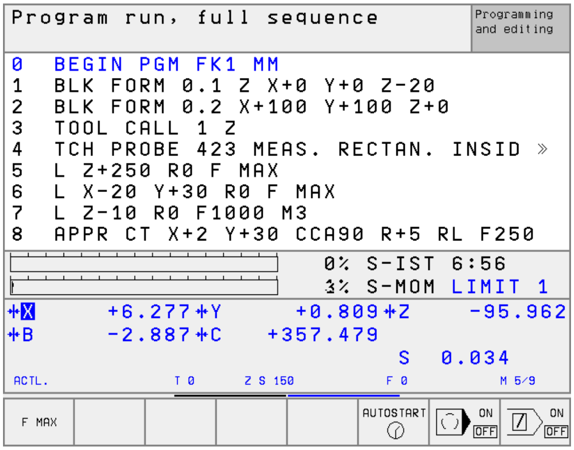
- Return to the contour after the machine axes were moved during a program interruption that was not performed with the INTERNAL STOP function.
- Return to the contour after a block scan with RESTORE POS AT N, for example after an interruption with INTERNAL STOP.
- **Additionally on the TNC 426 and TNC 430:** Depending on the machine, if the position of an axis has changed after the control loop has been opened during a program interruption:

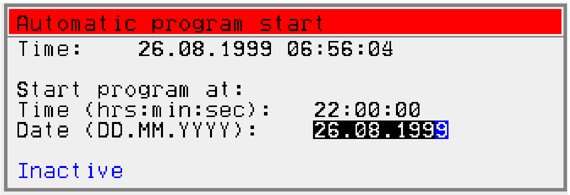
- ▶ To select a return to contour, press the RESTORE POSITION soft key.
- ▶ To move the axes in the sequence that the TNC suggests on the screen, press the machine START button.
- ▶ To move the axes in any sequence, press the soft keys RESTORE X, RESTORE Z, etc., and activate each axis with the machine START key.
- ▶ To resume machining, press the machine START key.

11.5 Automatic Program Start (not TNC 410)

Function

The TNC must be specially prepared by the machine tool builder for use of the automatic program start function. Refer to your machine manual.





In a Program Run operating mode, you can use the soft key AUTOSTART (see figure at upper right) to define a specific time at which the program that is currently active in this operating mode is to be started:

- Show the window for entering the starting time (see figure at center right).
- **`Time (h:min:sec):`** Time of day at which the program is to be started.
- **`Date (DD.MM.YYYY):`** Date at which the program is to be started.
- To activate the start, set the AUTOSTART soft key to ON.

11.6 Blockwise Transfer: Running Long Programs (not with TNC 426, TNC 430)

Function

Machine programs that require more memory space than is available in the TNC can be transferred "blockwise" from an external memory.

The program blocks are read in via data interface and are then deleted immediately after being executed. In this way programs of unlimited length can be executed.

The program may have a maximum of 20 G99 blocks. If you require more tools then use a tool table.

If the program contains a %... block, the called program must be stored in the TNC memory.

The program must not contain:

- Subprograms
- Program section repeats
- Function D15:PRINT

Blockwise program transfer

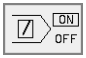
Configure the data interface with the MOD function

- Select the Program Run, Full Sequence mode or the Program Run, Single Block mode.
- Begin blockwise transfer: Press the BLOCKWISE TRANSFER soft key.
- Enter the program name and confirm your entry with the ENT key. The TNC reads in the selected program via data interface
- Start the part program with the machine START button.

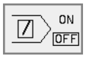
11.7 Optional block skip

Function

In a test run or program run, the TNC can skip over blocks that begin with a slash "/":



▶ To run or test the program without the blocks preceded by a slash, set the soft key to ON.



▶ To run or test the program with the blocks preceded by a slash, set the soft key to OFF.

> This function does not work for G99 blocks.
>
> After a power interruption the TNC returns to the most recently selected setting.

11.8 Optional Program Run Interruption

Function

The TNC optionally interrupts the program or test run at blocks containing M01. If you use M01 in the Program Run mode, the TNC does not switch off the spindle or coolant.

- Do not interrupt Program Run or Test Run at blocks containing M01: Set soft key to OFF

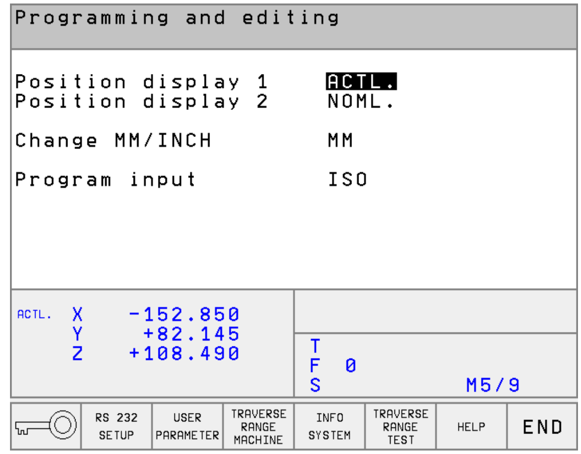
- Interrupt Program Run or Test Run at blocks containing M01: Set soft key to ON

12

MOD Functions

12.1 MOD functions

The MOD functions provide additional displays and input possibilities. The available MOD functions depend on the selected operating mode.



Selecting the MOD functions

Call the mode of operation in which you wish to change the MOD function.

MOD

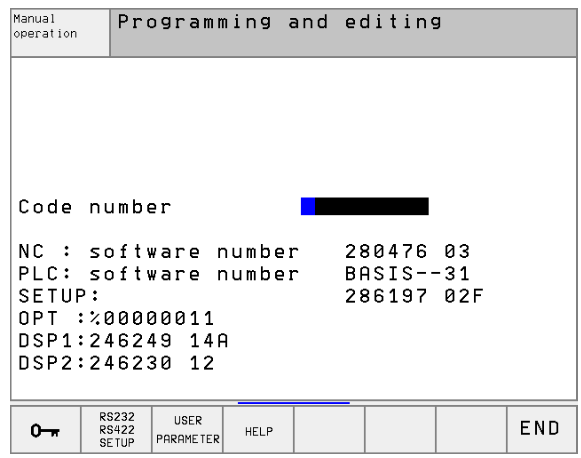
▶ To select the MOD functions, press the MOD key. Figure at upper right: MOD function on the TNC 410. Figures at center and lower right: MOD functions on the TNC 426, TNC 430 for Programming and Editing and Test Run; figure on next page: MOD function in a machine operating mode.

Changing the settings

▶ Select the desired MOD function in the displayed menu with the arrow keys.

There are three possibilities for changing a setting, depending on the function selected:

- Enter a numerical value directly, e.g. when determining traverse range limit.
- Change a setting by pressing the ENT key, e.g. when setting program input.
- Change a setting via a selection window (not TNC 410). If there are more than one possibilities for a particular setting available, you can superimpose a window listing all of the given possibilities by pressing the GOTO key. Select the desired setting directly by pressing the corresponding numerical key (to the left of the colon), or using the arrow keys and then confirming with ENT. If you don't want to change the setting, close the window again with END.



Exiting the MOD functions

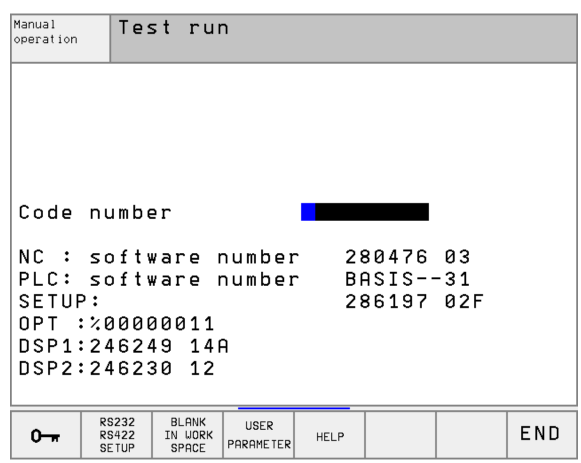
▶ Close the MOD functions with the END key or soft key.

Overview of MOD Functions TNC 426, TNC 430

Depending on the selected mode of operation, you can make the following changes:

Programming and Editing:

- Display software numbers
- Enter code number
- Set data interface
- Machine-specific user parameters (if provided)
- HELP files (if provided)

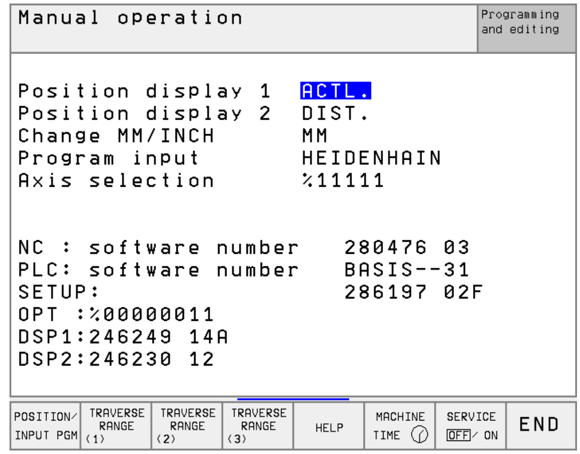


Test Run:

- Display software numbers
- Enter code number
- Setting the data interface
- Showing the Workpiece in the Working Space
- Machine-specific user parameters (if provided)
- Displaying HELP files (if provided)

In all other modes:

- Display software numbers
- Display code digits for installed options
- Select position display
- Unit of measurement (mm/inches)
- Programming language for MDI
- Select the axes for actual position capture
- Set the axis traverse limits
- Display the datums
- Displaying Operating Time
- HELP files (if provided)
- Activate Teleservice functions (if provided)



12.2 System Information (not TNC 426, TNC 430)

Function

You can use the soft key INFO SYSTEM to display the following information:

- Free program memory
- NC software number
- PLC software numbers are displayed on the TNC screen after the functions have been selected. Directly below them are the code numbers for the installed options (OPT:):
- Options (if present), e.g. digitizing

12.3 Software Numbers and Option Numbers (not TNC 410)

Function

The software numbers of the NC, PLC and the SETUP floppy disks appear in the TNC screen after the MOD functions have been selected. Directly below them are the code numbers for the installed options (OPT:):

No option OPT	00000000
Option for digitizing with triggering touch probe OPT	00000001
Option for digitizing with measuring touch probe OPT	00000011

12.4 Code Numbers

Function

Code numbers allow you to access various functions that are not always required for normal operation of the TNC.

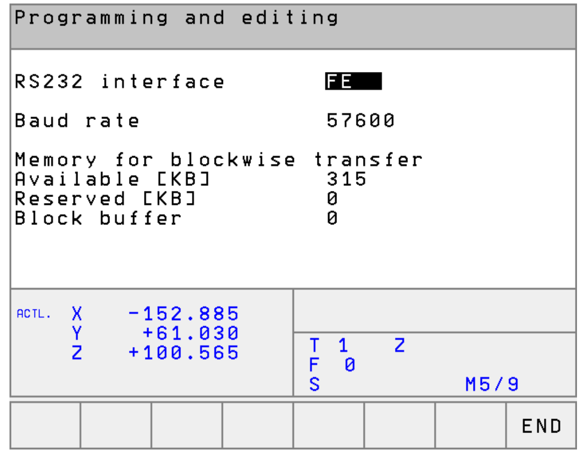
To enter the code number on the TNC 410, press the soft key with the key symbol. The TNC requires a code number for the following functions:

Function	Code number
Select user parameters	123
Enable special functions for Q-parameter programming	555343
Removing file protection (not TNC 426, TNC 430)	86357
Operating hours counter for (not TNC 426, TNC 430): CONTROL ON PROGRAM RUN SPINDLE ON	857282
Configuring an Ethernet card	NET123

12.5 Setting the Data Interface for the TNC 410

Selecting the setup menu

To setup the data interfaces, press the RS 232- / RS 422 - SETUP soft key to call a menu for setting the data interfaces:



Setting the OPERATING MODE of the external device

External device	Operating mode
PC with HEIDENHAIN data transfer software TNCremo or TNCremo NT	FE
HEIDENHAIN floppy disk units FE 401 and FE 401 FB	FE
Non-HEIDENHAIN devices such as punchers, PC without TNCremo	EXT1, EXT2
No data transfer; e.g. digitizing without position value capture, or working without an external device	none

Setting the BAUD RATE

You can set the BAUD RATE (data transfer speed) from 110 to 115 200 baud. The TNC stores an individual BAUD RATE for each operating mode (FE, EXT1 etc.).

Creating the memory for blockwise transfer

In order to be able to edit other programs while blockwise execution is in progress, you need to create a memory for blockwise transfer.

The TNC shows the available free memory space. The reserved memory space should be less than the total free memory space available.

Setting the block buffer

To ensure a continuous program run during blockwise transfer, the TNC needs a certain quantity of blocks stored in program memory.

In the block buffer you define how many NC blocks are read in through the data interface before the TNC begins the program run. The input value for the block buffer depends on the point intervals in the part program. For very small point intervals, enter a large block buffer. For large point intervals, enter a small block buffer. Proposed value: 1000

Data transfer between the TNC 410 and TNCremo

Ensure that:

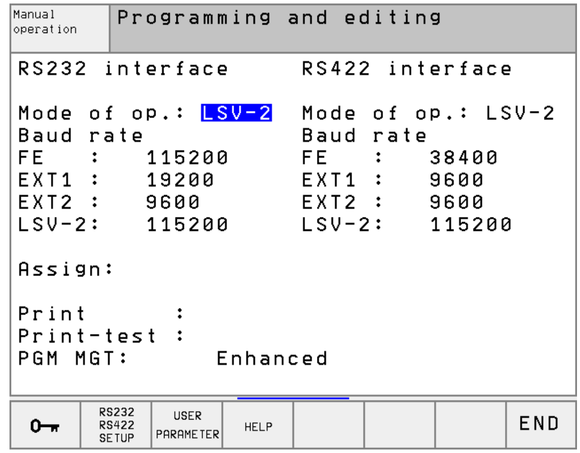
- The TNC is connected to the correct serial port on your PC.
- The data transfer speed set on the TNC for LSV2 operation is the same as that set on TNCremo.

Once you have started TNCremo, you will see a list of all of the files that are stored in the active directory on the left of the window. Using the menu items <Directory>, <Change>, you can change the active directory or select another directory. To start data transfer at the TNC (see “Data transfer to or from an external data medium” on page 69), select <Connect>, <File server>. TNCremo is now ready to receive data.

12.6 Setting the Data Interfaces for TNC 426, TNC 430

Selecting the setup menu

To setup the data interfaces, press the RS 232- / RS 422 - SETUP soft key to call a menu for setting the data interfaces:



Setting the RS-232 interface

The mode of operation and baud rates for the RS-232 interface are entered in the upper left of the screen.

Setting the RS-422 interface

The mode of operation and baud rates for the RS-422 interface are entered in the upper right of the screen.

Setting the OPERATING MODE of the external device

The functions “Transfer all files,” “Transfer selected file,” and “Transfer directory” are not available in the operating modes FE2 and EXT.

Setting the BAUD RATE

You can set the BAUD RATE (data transfer speed) from 110 to 115 200 baud.

External device	Operating mode	Symbol
PC with HEIDENHAIN software TNCremo for remote operation of the TNC	LSV2	
PC with HEIDENHAIN data transfer software TNCremo	FE1	
HEIDENHAIN floppy disk units FE 401 B FE 401 from prog. no. 230 626 03	 FE1 FE1	
HEIDENHAIN floppy disk unit FE 401 up to prog. no. 230 626 02	FE2	
Non-HEIDENHAIN devices such as punchers, PC without TNCremo	EXT1, EXT2	

Assign

This function sets the destination for the transferred data.

Applications:

- Transferring values with Q parameter function FN15
- Transferring values with Q parameter function FN16
- Path on the TNC's hard disk in which the digitized data are stored

The TNC mode of operation determines whether the PRINT or PRINT TEST function is used:

TNC mode of operation	Transfer function
Program Run, Single Block	PRINT
Program Run, Full Sequence	PRINT
Test Run	PRINT TEST

You can set PRINT and PRINT TEST as follows:

Function	Path
Output data via RS-232	RS232:\....
Output data via RS-422	RS422:\....
Save data to the TNC's hard disk	TNC:\....
Save data in directory in which the program with FN15/FN16 or the program with the digitizing cycles is located	- vacant -

File names

Data	Operating mode	File name
Surface data	Program Run	Defined in the RANGE cycle
Values with FN15	Program Run	%FN15RUN.A
Values with FN15	Test Run	%FN15SIM.A
Values with FN16	Program Run	%FN16RUN.A
Values with FN16	Test Run	%FN16SIM.A

Software for data transfer

For transfer of files to and from the TNC, we recommend using one of the HEIDENHAIN TNCremo data transfer software products for data transfer, such as TNCremo or TNCremoNT. With TNCremo/ TNCremoNT, data transfer is possible with all HEIDENHAIN controls via serial interface.

Please contact your HEIDENHAIN agent if you would like to receive the TNCremo or TNCremoNT data transfer software for a nominal fee.

System requirements for TNCremo:

- AT personal computer or compatible system
- Operating system MS-DOS/PC-DOS 3.00 or later, Windows 3.1, Windows for Workgroups 3.11, Windows NT 3.51, OS/2
- 640 KB working memory
- 1 MB free memory space on your hard disk
- One free serial interface
- A Microsoft-compatible mouse (for ease of operation, not essential)

System requirements for TNCremoNT:

- PC with 486 processor or higher
- Operating system Windows 95, Windows 98, Windows NT 4.0
- 16 MB working memory
- 5 MB free memory space on your hard disk
- One free serial interface or connection to the TCP/IP network on TNCs with Ethernet card

Installation under Windows

- ▶ Start the SETUP.EXE installation program with the file manager (Explorer).
- ▶ Follow the setup program instructions.

Starting TNCremo under Windows 3.1, 3.11 and NT 3.51

Windows 3.1, 3.11, NT 3.51:

- ▶ Double-click on the icon in the program group HEIDENHAIN Applications.

When you start TNCremo for the first time, you will be asked for the type of control you have connected, the interface (COM1 or COM2) and the data transfer speed. Enter the necessary information.

Starting TNCremoNT under Windows 95, Windows 98 and NT 4.0

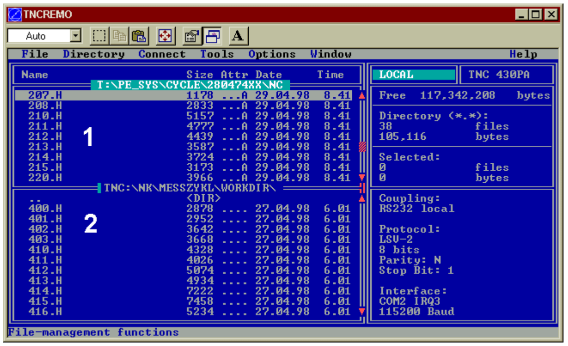
- ▶ Click <Start>, <Programs>, <HEIDENHAIN Applications>, <TNCremoNT>

When you start TNCremoNT for the first time, TNCremoNT automatically tries to set up a connection with the TNC.

Data transfer between the TNC and TNCremo

Ensure that:

- The TNC is connected to the correct serial port on your PC.
- The operating mode of the interface is set to **LSV2** on the TNC.
- The data transfer speed set on the TNC for LSV2 operation is the same as that set on TNCremo.



Once you have started TNCremo, you will see a list of all of the files that are stored in the active directory on the left side of the main window **1** . Using the menu items <Directory>, <Change>, you can change the active directory or select another directory on your PC.

If you want to control data transfer from the PC, establish the connection with your PC in the following way:

- Select <Connect>, <Link>. TNCremo now receives the file and directory structure from the TNC and displays this at the bottom left of the main window **2**.
- To transfer a file from the TNC to the PC, select the file in the TNC window (highlighted with a mouse click) and activate the functions <File> <Transfer>.
- To transfer a file from the PC to the TNC, select the file in the PC window (highlighted with a mouse click) and activate the functions <File> <Transfer>.

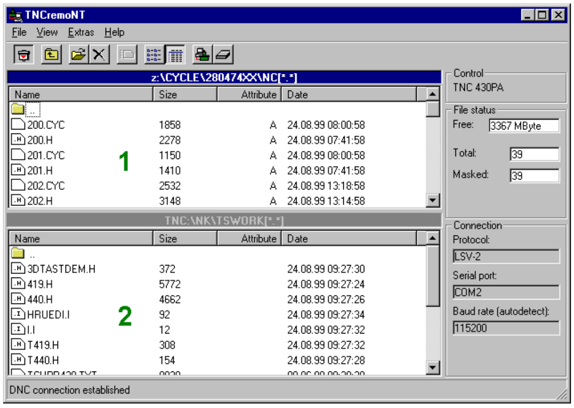
If you want to control data transfer from the TNC, establish the connection with your PC in the following way:

- Select <Connect>, <File server (LSV2)>. TNCremo is now in server mode. It can receive data from the TNC and send data to the TNC.
- You can now call the file management functions on the TNC by pressing the key PGM MGT (see “Data transfer to or from an external data medium” on page 62) and transfer the desired files.

End TNCremo

Select the menu items <File>, <Exit>, or press the key combination ALT+X.

Refer also to the TNCremoNT help texts where all of the functions are explained in more detail.



Data transfer between the TNC and TNCremoNT

Ensure that:

- The TNC is connected to the correct serial port on your PC or to the network, respectively.
- The operating mode of the interface is set to **LSV2** on the TNC.

Once you have started TNCremoNT, you will see a list of all files that are stored in the active directory in the upper section of the main window **1** . Using the menu items <File>, <Change directory>, you can change the active directory or select another directory on your PC.

If you want to control data transfer from the PC, establish the connection with your PC in the following way:

- Select <File>, <Setup connection>. TNCremoNT now receives the file and directory structure from the TNC and displays this at the bottom left of the main window **2**.
- To transfer a file from the TNC to the PC, select the file in the TNC window with a mouse click and drag and drop the highlighted file into the PC window **1**.
- To transfer a file from the PC to the TNC, select the file in the PC window with a mouse click and drag and drop the highlighted file into the TNC window **2**.

If you want to control data transfer from the TNC, establish the connection with your PC in the following way:

- Select <Extras>, <TNCserver>. TNCremoNT is now in server mode. It can receive data from the TNC and send data to the TNC.
- You can now call the file management functions on the TNC by pressing the key PGM MGT (see “Data transfer to or from an external data medium” on page 62) and transfer the desired files.

End TNCremoNT

Select the menu items <File>, <Exit>.

Refer also to the TNCremoNT help texts where all of the functions are explained in more detail.

12.7 Ethernet Interface (not TNC 410)

Introduction

As an option, you can equip the TNC with an Ethernet card to connect the control as a client in your network. The TNC transmits data through the Ethernet card in accordance with the Transmission Control Protocol/Internet Protocol (TCP/IP) family of protocols and with the aid of the Network File System (NFS). Since TCP/IP and NFS are implemented in UNIX systems, you can usually connect the TNC in the UNIX world without any additional software.

The PC world with Microsoft operating systems, however, also works with TCP/IP, but not with NFS. You will therefore need additional software to connect the TNC to a PC network. For the operating systems Windows 95, Windows 98 and Windows NT 4.0, HEIDENHAIN recommends the network software **CimcoNFS for HEIDENHAIN** which you can order separately or together with the Ethernet card for the TNC.:

Item	HEIDENHAIN ID number
Only software CimcoNFS for HEIDENHAIN	339 737-01
Ethernet card and software CimcoNFS for HEIDENHAIN	293 890-73

Installing an Ethernet card

Switch-off the TNC and the machine before you install an Ethernet card!

Read the installation instruction supplied with the Ethernet card!

Connection possibilities

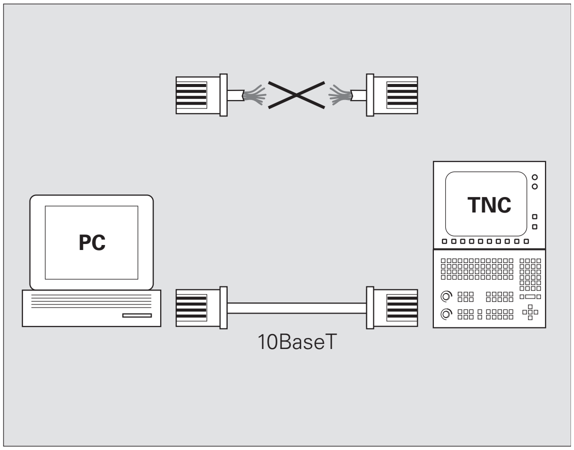
You can connect the Ethernet card in your TNC to your network through the RJ45 connection (X26, 10BaseT). The connection is metallically isolated from the control electronics.

RJ45 connection X26 (10BaseT)

For a 10BaseT connection you need a twisted-pair cable to connect the TNC to your network.

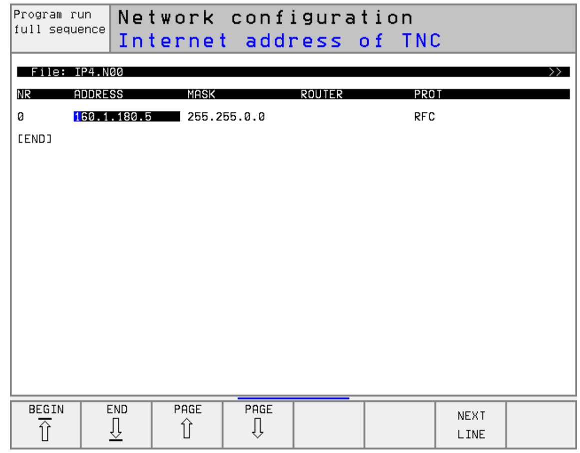
For unshielded cables, the maximum cable length between the TNC and a node is 100 meters (329 ft). For shielded cables, it is 400 meters (1300 ft).

If you connect the TNC directly with a PC you must use a transposed cable.



Configuring the TNC

> Make sure that the person configuring your TNC is a network specialist.



- In the Programming and Editing mode of operation, press the MOD key. Enter the code word NET123. The TNC will then display the main screen for network configuration.

General network settings

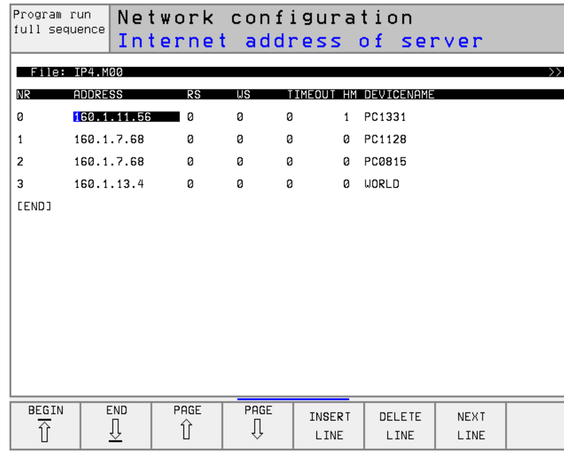
- Press the DEFINE NET soft key to enter the general network settings and enter the following information:

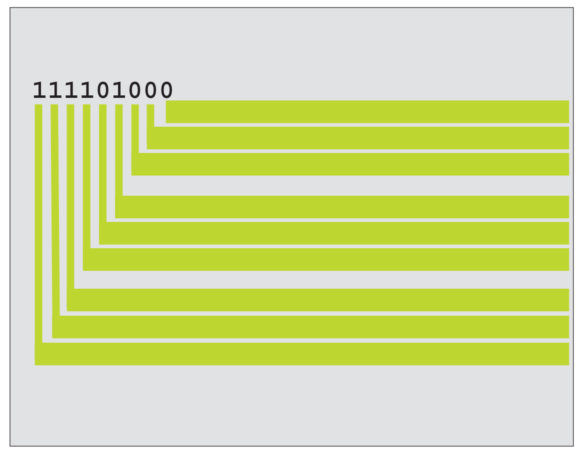
Setting	Meaning
ADDRESS	Address that your network manager must assign to the TNC. Input: four decimal numbers separated by points, e.g. 160.1.180.20
MASK	The SUBNET MASK for expanding the number of available addresses within your network. Input: four decimal numbers separated by points. Ask your network manager for the number of your subnet mask, e.g. 255.255.0.0.
ROUTER	Internet address of your default router. Enter the Internet address only if your network consists of several parts. Input: four decimal numbers separated by points. Ask your network manager for your address, e.g. 160.2.0.2.
PROT	Definition of the transmission protocol. RFC: Transmission protocol according to RFC 894 IEEE: Transmission protocol according to IEEE 802.2/802.3
HW	Definition of the connection used 10BASET: for use of 10BaseT
HOST	Name, under which the TNC identifies itself in the network. If you are using a host name, you must enter the “Fully Qualified Hostname” here. If you do not enter a name here, the TNC uses the so-called null authentication. The UID, GID, DCM and FCM settings specific to the device (see next page), are then ignored by the TNC.

Network settings specific to the device

- Press the soft key DEFINE MOUNT to enter the network settings for a specific device. You can define any number of network settings, but you can manage only seven at one time.

Setting	Meaning
ADDRESS	Address of your server. Input: four decimal numbers separated by points. Ask your network manager for the number of your address. e.g. 160.1.13.4.
RS	Packet size in bytes for data reception. Input range: 512 to 4096. Input 0: The TNC uses the optimal packet size as reported by the server.
WS	Packet size in bytes for data transmission. Input range: 512 to 4096. Input 0: The TNC uses the optimal packet size as reported by the server.
TIMEOUT	Time in ms, after which the TNC repeats a Remote Procedure Call. Input range: 0 to 100 000. Standard input: 700, which corresponds to a TIMEOUT of 700 milliseconds. Use higher values only if the TNC must communicate with the server through several routers. Ask your network manager for the proper timeout setting.
HM	Definition of whether the TNC should repeat the Remote Procedure Call until the NFS server answers. 0: Always repeat the Remote Procedure Call 1: Do not repeat the Remote Procedure Call
DEVICENAME	Name that the TNC shows in the file manager for a connected device.
PATH	Directory of the NFS server that you wish to connect to the TNC. Be sure to differentiate between small and capital letters when entering the path.
UID	Definition of the User Identification under which you access files in the network. Ask your network manager for the proper timeout setting.
GID	Definition of the group identification with which you access files in the network. Ask your network manager for the proper timeout setting.
DCM	Here you enter the rights of access to the NFS server (see figure at center right). Enter a binary coded value. Example: 111101000 0: Access not permitted 1: Access permitted





Setting	Meaning
DCM	Here you enter the rights of access to files on the NFS server (see figure at upper right). Enter a binary coded value. Example: 111101000 0: Access not permitted 1: Access permitted
AM	Definition of whether the TNC upon switch-on should automatically connect with the network. 0: Do not automatically connect 1: Connect automatically

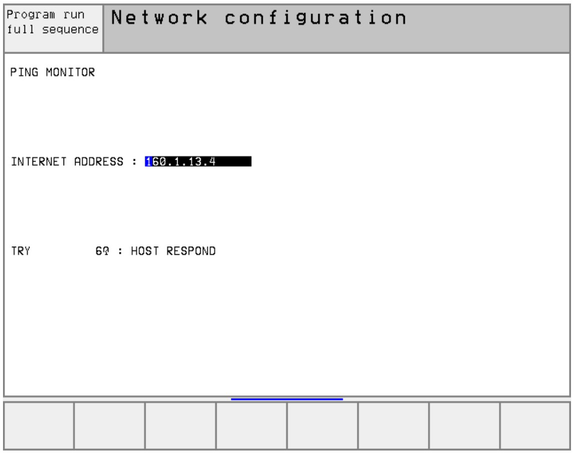
Defining the network printer

- Press the DEFINE PRINT soft key if you wish to print the files on the network printer directly from the TNC.

Setting	Meaning
ADDRESS	Address of your server. Input: four decimal numbers separated by points. Ask your network manager for the number of your address. e.g. 160.1.13.4.
DEVICE NAME	Name of printer that the TNC shows when the PRINT soft key is pressed, see "Expanded File Management TNC 426, TNC 430," page 53
PRINTER NAME	Name of the printer in your network. Ask your network manager.

Checking the network connection

- Press the PING soft key.
- Enter the Internet address of the device with which you wish to check the connection, and confirm your entry with ENT. The TNC transmits data packets until you exit the test monitor by pressing the END key.



In the TRY line the TNC shows the number of data packets that were transmitted to the previously defined addressee. Behind the number of transmitted data packets the TNC shows the status:

Status display	Meaning
HOST RESPOND	Data packet was received again, connection is OK.
TIMEOUT	Data packet was not received, check the connection.
CAN NOT ROUTE	Data packet could not be transmitted. Check the Internet address of the server and of the router to the TNC.

Displaying the error log

- Press the SHOW ERROR soft key if you would like to see the error log. Here the TNC records all errors that have occurred in the network since the TNC was last switched on.

The listed error messages are divided into two categories:

Warnings are indicated with (W). Warnings occur when the TNC was able to establish the network connection, but had to correct settings in order to do so.

Error messages are indicated with (E). Error messages occur when the TNC was unable to establish a network connection.

Error message	Cause
LL: (W) CONNECTION xxxxx UNKNOWN USING DEFAULT 10BASET	The name you entered in DEFINE NET, HW was incorrect
LL: (E) PROTOCOL xxxxx UNKNOWN	The name you entered in DEFINE NET, PROT was incorrect
IP4: (E) INTERFACE NOT PRESENT	The TNC was unable to find an Ethernet card.
IP4: (E) INTERNETADDRESS NOT VALID	You used an invalid Internet address for the TNC.
IP4: (E) SUBNETMASK NOT VALID	The SUBNET MASK does not match the Internet address of the TNC.
IP4: (E) SUBNETMASK OR HOST ID NOT VALID	You used an invalid Internet address for the TNC, or you entered an incorrect SUBNET MASK, or you set all of the HostID bits to 0 (1)
IP4: (E) SUBNETMASK OR SUBNET ID NOT VALID	All bits of the SUBNET ID are 0 or 1
IP4: (E) DEFAULTROUTERADRESS NOT VALID	You used an invalid Internet address for the router.
IP4: (E) CAN NOT USE DEFAULTROUTER	The default router does not have the same net ID or subnet ID as the TNC.
IP4: (E) I AM NOT A ROUTER	You defined the TNC as a router.
MOUNT: <Device name> (E) DEVICENAME NOT VALID	The device name is either too long or it contains illegal characters.
MOUNT: <Device name> (E) DEVICENAME ALREADY ASSIGNED	You have already defined a device with this name.
MOUNT: <Device name> (E) DEVICETABLE OVERFLOW	You have attempted to connect more than seven network drives to the TNC.
NFS2: <Device name> (W) READSIZE SMALLER THEN x SET TO x	The value that you entered for DEFINE MOUNT, RS is too small. The TNC sets RS to 512 bytes.
NFS2: <Device name> (W) READSIZE LARGER THEN x SET TO x	The value that you entered for DEFINE MOUNT, RS is too large. The TNC sets RS to 4096 bytes.
NFS2: <Device name> (W) WRITESIZE SMALLER THEN x SET TO x	The value that you entered for DEFINE MOUNT, WS is too small. The TNC sets WS to 512 bytes.

Error message	Cause
NFS2: <Device name> (W) WRITESIZE LARGER THEN x SET TO x	The value that you entered for DEFINE MOUNT, WS is too large. The TNC sets WS to 4096 bytes.
NFS2: <Device name> (E) MOUNTPATH TO LONG	The name you entered in DEFINE MOUNT, PATH is too long.
NFS2: <Device name> (E) NOT ENOUGH MEMORY	At the moment there is too little main memory available to establish a network connection.
NFS2: <Device name> (E) HOSTNAME TO LONG	The name you entered in DEFINE NET, HOST is too long.
NFS2: <Device name> (E) CAN NOT OPEN PORT	The TNC cannot open the port required to establish the network connection.
NFS2: <Device name> (E) ERROR FROM PORTMAPPER	The TNC has received implausible data from the portmapper.
NFS2: <Device name> (E) ERROR FROM MOUNTSERVER	The TNC has received implausible data from the mountserver.
NFS2: <Device name> (E) CANT GET ROOTDIRECTORY	The mount server does not permit a connection with the directory defined in DEFINE MOUNT, PATH.
NFS2: <Device name> (E) UID OR GID 0 NOT ALLOWED	You entered 0 for DEFINE MOUNT, UID or GID 0. The input value 0 is reserved for the system administrator.

12.8 Configuring PGM MGT (not TNC 410)

Function

With this function you can determine the features of the file manager:

- Standard: Simple file management without directory display
- Expanded range: File management with additional functions and directory display

Note: see "Standard File Management TNC 426, TNC 430," page 45, and see "Expanded File Management TNC 426, TNC 430," page 53.

Changing the setting

- Select the file manager in the Programming and Editing mode of operation: press the PGM MGT key
- Select the MOD function: Press the MOD key.
- Select the PGM MGT setting: using the arrow keys, move the highlight onto the PGM MGT setting and use the ENT key to switch between STANDARD and ENHANCED.

12.9 Machine-Specific User Parameters

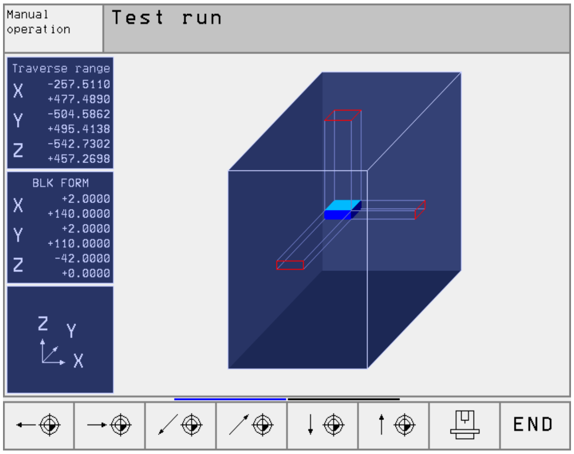
Function

To enable you to set machine-specific functions, your machine tool builder can define up to 16 machine parameters as user parameters.

This function is not available on all TNCs. Refer to your machine manual.

12.10 Showing the Workpiece in the Working Space (not TNC 410)

Function



This MOD function enables you to graphically check the position of the workpiece blank in the machine's working space and to activate work space monitoring in the Test Run mode of operation. This function is activated with the BLANK IN WORK SPACE soft key.

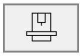
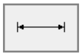
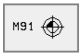
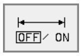
The TNC displays a cuboid for the working space. Its dimensions are shown in the "Traverse range" window. The TNC takes the dimensions for the working space from the machine parameters for the active traverse range. Since the traverse range is defined in the reference system of the machine, the datum of the cuboid is also the machine datum. You can see the position of the machine datum in the cuboid by pressing the soft key M91 in the 2nd soft-key row.

Another cuboid represents the blank form. The TNC takes its dimensions from the workpiece blank definition in the selected program. The workpiece cuboid defines the coordinate system for input. Its datum lies within the cuboid. You can see in the cuboid the position of the datum for input by pressing the corresponding soft key in the 2nd soft-key row.

For a test run it normally does not matter where the workpiece blank is located within the working space. However, if you test programs that contain movements with M91 or M92, you must graphically shift the workpiece blank to prevent contour damage. Use the soft keys shown in the table at right.

You can also activate the working-space monitor for the Test Run mode in order to test the program with the current datum and the active traverse ranges (see table below, last line).

Function	Soft key
Move workpiece blank to the left	
Move workpiece blank to the right	
Move workpiece blank forward	
Move workpiece blank backward	

Function	Soft key
Move workpiece blank upward	
Move workpiece blank downward	
Show workpiece blank referenced to the set datum	
Show the entire traversing range referenced to the displayed workpiece blank	
Show the machine datum in the working space	
Show a position determined by the machine tool builder (e.g. tool change position) in the working space	
Show the workpiece datum in the working space	
Enable (ON) or disable (OFF) work space monitoring	

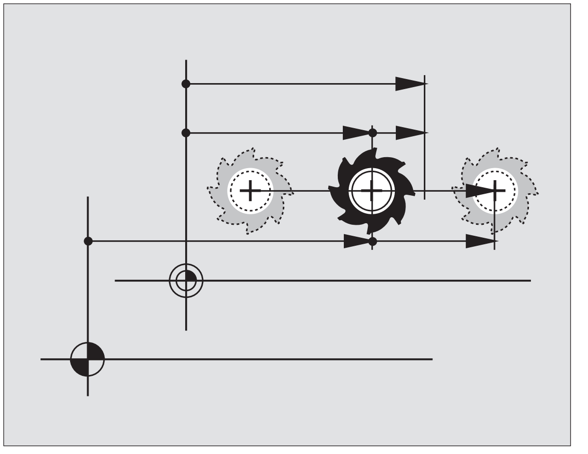
12.11 Position Display Types

Function

In the Manual Operation mode and in the Program Run modes of operation, you can select the type of coordinates to be displayed.

The figure at right shows the different tool positions:

- Starting position
- Target position of the tool
- Workpiece datum
- Machine datum



The TNC position displays can show the following coordinates:

Function	Display
Nominal position: the value presently commanded by the TNC	`NOML.`
Actual position; current tool position	`ACTL.`
Reference position; the actual position relative to the machine datum	`REF`
Distance remaining to the programmed position; difference between actual and target positions	`DIST.`
Servo lag: difference between nominal and actual positions (following error)	`LAG`
Deflection of the measuring touch probe	`DEFL.`
Traverses that were carried out with handwheel superpositioning (M118) (only position display 2, not TNC 410)	`M118`

With the MOD function Position display 1, you can select the position display in the status display.

With Position display 2, you can select the position display in the additional status display.

12.12 Unit of Measurement

Function

This MOD function determines whether the coordinates are displayed in millimeters (metric system) or inches.

- To select the metric system (e.g. X = 15.789 mm) set the Change mm/inches function to mm. The value is displayed to 3 decimal places.
- To select the inch system (e.g. X = 0.6216 inch) set the Change mm/inches function to inches. The value is displayed to 4 decimal places.

If you activate inch display, the TNC shows the feed rate in inch/min. In an inch program you must enter the feed rate large by a factor of 10.

12.13 Select the Programming Language for $MDI

Function

The Program input MOD function lets you decide whether to program the $MDI file in HEIDENHAIN conversational dialog or in ISO format.

- To program the $MDI.H file in conversational dialog, set the Program input function to HEIDENHAIN
- To program the $MDI.I file according to ISO, set the Program input function to ISO

12.14 Selecting the Axes for Generating L Blocks (not TNC 410)

Function

This function is only available with conversational dialog programming.

The axis selection input field enables you to define the current tool position coordinates that are transferred to an L block. To generate a separate L block, press the ACTUAL-POSITION-CAPTURE soft key. The axes are selected by bit-oriented definition similar to programming the machine parameters:

Axis selection %11111 Transfer the X, Y, Z, IV and V axes

Axis selection %01111 Transfer the X, Y, Z and IV axes

Axis selection %00111 Transfer the X, Y and Z axes

Axis selection %00011 Transfer the X and Y axes

Axis selection %00001 Transfer the X axis

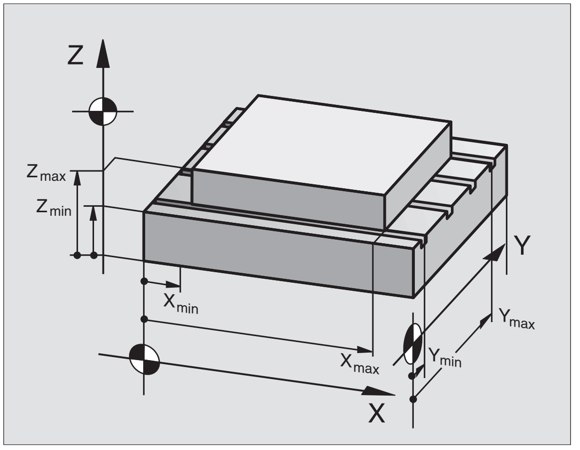
12.15 Enter the Axis Traverse Limits, Datum Display

Function

The AXIS LIMIT MOD function allows you to set limits to axis traverse within the machine's actual working envelope.

Possible application: Protecting an indexing fixture against tool collision.

The maximum range of traverse of the machine tool is defined by software limit switches. This range can be additionally limited through the TRAVERSE RANGE MOD function. With this function, you can enter the maximum and minimum traverse positions for each axis, referenced to the machine datum. If several traverse ranges are possible on your machine, you can set the limits for each range separately using the soft keys TRAVERSE RANGE (1) to TRAVERSE RANGE (3).

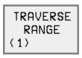


Working without additional traverse limits

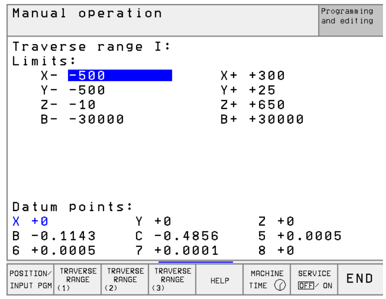
To allow a machine axis to use its full range of traverse, enter the maximum traverse of the TNC (+/- 99999 mm) as the TRAVERSE RANGE.

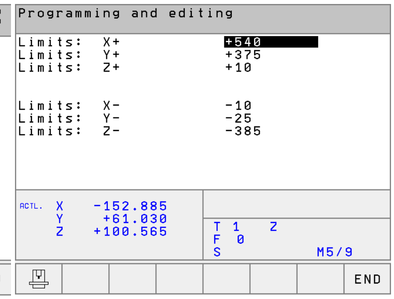
Find and enter the maximum traverse

- Set the Position display MOD function to **REF**.
- Move the spindle to the positive and negative end positions of the X, Y and Z axes.
- Write down the values, including the algebraic sign.
- To select the MOD functions, press the MOD key.



- Enter the limits for axis traverse: Press the TRAVERSE RANGE soft key and enter the values that you wrote down as limits in the corresponding axes
- To exit the MOD functions, press the END soft key.





> The tool radius is not automatically compensated in the axis traverse limit value.
>
> The traverse range limits and software limit switches become active as soon as the reference points are traversed.

Datum display

The values shown at the lower left of the screen are the manually set datums referenced to the machine datum. They cannot be changed in the menu.

Axis traverse limits for test run (not TNC 426, TNC 430)

It is possible to define a separate "traverse range" (switch soft-key row as required) for the test run and programming graphics after you have activated the MOD function.

In addition to the axis traverse limits, you can also define the position of the workpiece datum referenced to the machine datum.

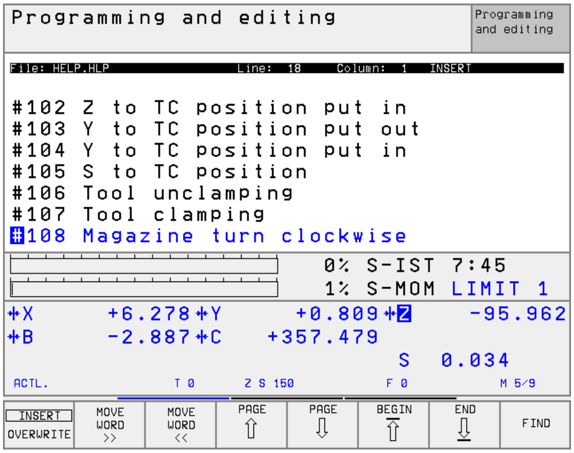
12.16 The HELP Function

Function

> The HELP function is not available on every machine. Your machine tool builder can provide you with further information on this feature.

The HELP function can aid you in situations in which you need clear instructions before you can continue (for example, to retract the tool after an interruption of power). The miscellaneous functions may also be explained in a help file.

The TNC 426, TNC 430 offers several help files, which you can select by means of the file management. The figure at upper right shows the screen display of a help file on the TNC 426, TNC 430.



Selecting and executing a HELP function

- To select the MOD function, press the MOD key.

HELP

- Select the HELP function with the HELP soft key.
- On the TNC 426, TNC 430, call the file manager (PGM MGT key) and select a different help file, if necessary.
- Use the up and down arrow keys to select a line in the HELP file, which is marked with an #.
- Use the NC start key to execute the selected HELP function.

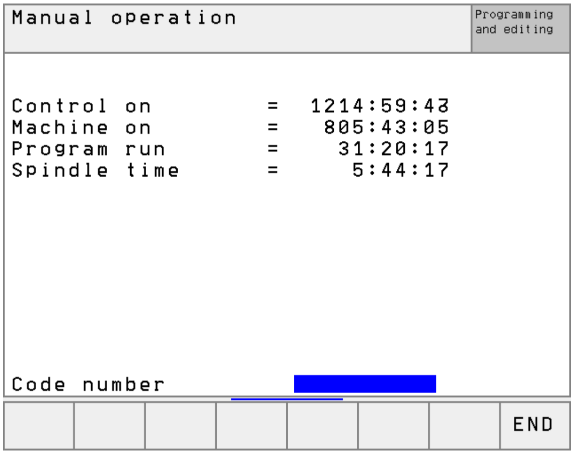
12.17 Operating Time (via Code Number for TNC 410)

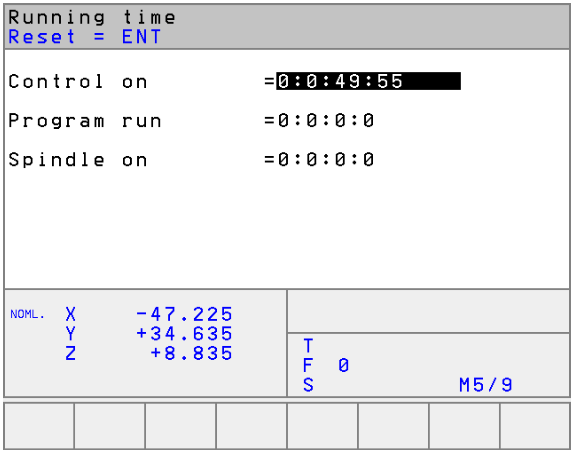
Function

> The machine tool builder can provide further operating time displays. The machine tool manual provides further information.

The MACHINE TIME soft key enables you to show different operating time displays:

Operating time	Meaning
Control ON	Operating time of the control since commissioning
Machine ON	Operating time of the machine tool since commissioning
Program Run	Duration of controlled operation since commissioning

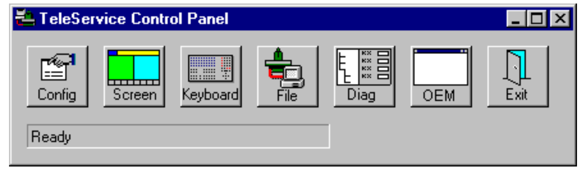




12.18 Teleservice (not TNC 410)

Function

> The Teleservice functions are enabled and adapted by the machine tool builder. The machine tool manual provides further information.
> The TNC provides two soft keys for the Teleservice, making it possible to configure two different service agencies.



The TNC allows you to carry out Teleservice. To be able to use this feature, your TNC should be equipped with an Ethernet card which achieves a higher data transfer rate than the serial RS232-C interface.

With the HEIDENHAIN TeleService software, your machine tool builder can then establish a connection to the TNC via an ISDN modem and carry out diagnostics. The following functions are available:

- On-line screen transfer
- Polling of machine states
- Data transfer
- Remote control of the TNC

Generally, a connection via the Internet would also be possible. First tests have shown, however, that the transfer rate that can currently be achieved is not sufficient due to the high degree of utilization of the networks.

Calling/Exiting Teleservice

- Select any machine mode of operation.
- To select the MOD function, press the MOD key.

SERVICE OFF / ON

- Establish a connection to the service agency: Set the SERVICE or SUPPORT soft key to ON. The TNC breaks the connection automatically if no new data have been transferred for a time set by the machine tool builder (default: 15 min).
- To break the connection to the service agency: Set the SERVICE or SUPPORT soft key to OFF. The TNC terminates the connection after approx. one minute.

12.19 External Access (not TNC 410)

Function

The machine tool builder can configure Teleservice settings with the LSV-2 interface. The machine tool manual provides further information.

The soft key SERVICE can be used to grant or restrict access through the LSV-2 interface.

With an entry in the configuration file TNC.SYS you can protect a directory and its subdirectories with a password. The password is requested when data from this directory is accessed from the LSV-2 interface. Enter the path and password for external access in the configuration file TNC.SYS.

The TNC.SYS file must be stored in the root directory TNC:\.

If you only supply one entry for the password, then the entire drive TNC:\ is protected.

You should use the updated versions of the HEIDENHAIN software TNCremo or TNCremoNT to transfer the data.

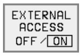
Entries in TNC.SYS	Meaning
REMOTE.TNCPASSWORD=	Password for LSV-2 access
REMOTE.TNCPRIVATEPATH=	Path to be protected

Example of TNC.SYS

```
REMOTE.TNCPASSWORD=KR1402
REMOTE.TNCPRIVATEPATH=TNC:\RK
```

Permitting/Restricting external access

- Select any machine mode of operation.
- To select the MOD function, press the MOD key.



- Permit a connection to the TNC: Set the EXTERNAL ACCESS soft key to ON. The TNC will then permit data access through the LSV-2 interface. The password is requested when a directory that was entered in the configuration file TNC.SYS is accessed.
- Block connections to the TNC: Set the EXTERNAL ACCESS soft key to OFF. The TNC will then block access through the LSV-2 interface.

13

Tables and Overviews

13.1 General User Parameters

General user parameters are machine parameters affecting TNC settings that the user may want to change in accordance with his requirements.

Some examples of user parameters are:

- Dialog language
- Interface behavior
- Traversing speeds
- Sequence of machining
- Effect of overrides

Input possibilities for machine parameters

Machine parameters can be programmed as

- **Decimal numbers**
  Enter only the number
- **Pure binary numbers**
  Enter a percent sign (%) before the number
- **Hexadecimal numbers**
  Enter a dollar sign ($) before the number

Example:

Instead of the decimal number 27 you can also enter the binary number %11011 or the hexadecimal number $1B.

The individual machine parameters can be entered in the different number systems.

Some machine parameters have more than one function. The input value for these machine parameters is the sum of the individual values. For these machine parameters the individual values are preceded by a plus sign.

Selecting general user parameters

General user parameters are selected with code number 123 in the MOD functions.

> The MOD functions also include machine-specific user parameters.

External data transfer	
Integrating TNC interfaces EXT1 (5020.0) and EXT2 (5020.1) to an external device	**MP5020.x** 7 data bits (ASCII code, 8th bit = parity): **+0** 8 data bits (ASCII code, 9th bit = parity): **+1** Block Check Character (BCC) any: **+0** Block Check Character (BCC) control character not permitted: **+2** Transmission stop through RTS active: **+4** Transmission stop through RTS inactive: **+0** Transmission stop through DC3 active: **+8** Transmission stop through DC3 inactive: **+0** Character parity even: **+0** Character parity odd: **+16** Character parity not desired: **+0** Character parity desired: **+32** 11/2 stop bits: **+0** 2 stop bits: **+64** 1 stop bit: **+128** 1 stop bits: **+192** Example: Use the following setting to adjust the TNC interface EXT2 (MP 5020.1) to an external non-HEIDENHAIN device: 8 data bits, any BCC, transmission stop through DC3, even character parity, character parity desired, 2 stop bits Input for **MP 5020.1**: 1+0+8+0+32+64 = **105**
Interface type for EXT1 (5030.0) and EXT2 (5030.1)	**MP5030.x** Standard transmission: **0** Interface for blockwise transfer: **1**

3-D touch probes and digitizing	
Select touch probe (only with option for digitizing with measuring touch probe; not TNC 410)	**MP6200** Triggering touch probe: **0** Measuring touch probe: **1**
Select signal transmission	**MP6010** Touch probe with cable transmission: **0** Touch probe with infrared transmission: **1**
Probing feed rate for triggering touch probes	**MP6120** **1** to **3 000** [mm/min]
Maximum traverse to first probe point	**MP6130** **0.001** to **99 999.9999** [mm]
Safety clearance to probing point during automatic measurement	**MP6140** **0.001** to **99 999.9999** [mm]

3-D touch probes and digitizing	
Rapid traverse for triggering touch probes	**MP6150** **1** to **300 000** [mm/min]
Measure center misalignment of the stylus when calibrating a triggering touch probe	**MP6160** No 180° rotation of the 3-D touch probe during calibration: **0** M function for 180° rotation of the touch probe during calibration: **1** to **999**
M function for orienting the infrared sensor before each measuring cycle (not TNC 410)	**MP6161** Function inactive: **0** Orientation directly through the NC: **-1** M function for orienting the touch probe: **1** to **999**
Angle of orientation for the infrared sensor (not TNC 410)	**MP6162** **0** to **359.9999** [°]
Difference between the current angle of orientation and the angle of orientation set in MP 6162; when the entered difference is reached, an oriented spindle stop is to be carried out (not TNC 410)	**MP6163** **0** to **3.0000** [°]
Automatically orient the infrared sensor before probing to the programmed probing direction	**MP6165** Function inactive: **0** Orient infrared sensor: **1**
Multiple measurement for programmable probe functions (not TNC 410)	**MP6170** **1** to **3**
Confidence interval for multiple measurement (not TNC 410)	**MP6171** **0.001** to **0.999** [mm]
Automatic calibration cycle: Center of the calibration ring in the X-axis referenced to the machine datum (not TNC 410)	**MP6180.0 (traverse range 1) to MP6180.2 (traverse range3)** **0** to **99 999.9999** [mm]
Automatic calibration cycle: Center of the calibration ring in the Y-axis referenced to the machine datum (not TNC 410)	**MP6181.x (traverse range 1) to MP6181.2 (traverse range3)** **0** to **99 999.9999** [mm]
Automatic calibration cycle: Upper edge of the calibration ring in the Z-axis referenced to the machine datum (not TNC 410)	**MP6182.x (traverse range 1) to MP6182.2 (traverse range3)** **0** to **99 999.9999** [mm]
Automatic calibration cycle: distance below the upper edge of the ring where the calibration is carried out by the TNC	**MP6185.x (traverse range 1) to MP6185.2 (traverse range 3)** **0.1** to **99 999.9999** [mm]
Infeed of the stylus when digitizing with the measuring touch probe (not TNC 410)	**MP6310** **0.1** to **2.0000** [mm] (recommended input value: 1mm)
Measure center misalignment of the stylus when calibrating a measuring touch probe (not TNC 410)	**MP6321** Measure center misalignment: **0** Do not measure center misalignment: **1**

3-D touch probes and digitizing	
Assign touch probe axis to machine axis for a measuring touch probe (not TNC 410) **Note**: Ensure that the touch probe axes are correctly assigned to the machine axes. Wrong assignment could lead to a stylus break.	**MP6322.0** Machine **X** axis parallel to touch probe axis X: **0**, Y: **1**, Z: **2** **MP6322.1** Machine **Y** axis parallel to touch probe axis X: **0**, Y: **1**, Z: **2** **MP6322.2** Machine **Z** axis parallel to touch probe axis X: **0**, Y: **1**, Z: **2**
Maximum stylus deflection of the measuring touch probe (not TNC 410)	**MP6330** **0.1** to **4.0000** [mm]
Feed rate for positioning measuring touch probes at MIN point and approaching the contour (not TNC 410)	**MP6350** **1** to **3 000** [mm/min]
Probe rate for measuring touch probe (not TNC 410)	**MP6360** **1** to **3 000** [mm/min]
Rapid traverse for measuring touch probes in the probe cycle (not TNC 410)	**MP6361** **10** to **3 000** [mm/min]
Feed rate reduction when the stylus of a measuring touch probe is deflected to the side (not TNC 410) The TNC decreases the feed rate according to a preset characteristic curve. The minimum input value is 10% of the programmed digitizing feed rate.	**MP6362** Feed rate reduction not active: **0** Feed rate reduction active: **1**
Radial acceleration during digitizing for measuring touch probe (not TNC 410) MP6370 enables you to limit the feed rate of the TNC for circular movements during digitizing. Circular movements are caused, for example, by sharp changes of direction. As long as the programmed digitizing feed rate is less than the feed rate calculated with MP6370, the TNC will move at the programmed feed rate. Determine the appropriate value for your requirements by trial and error.	**MP6370** **0.001** to **5.000** [m/s^2] (recommended input value: 0.1)
Target window for digitizing contour lines with a measuring touch probe (not TNC 410) When you are digitizing contour lines the individual contour lines do not end exactly in their starting points. With machine parameter MP6390 you can define a square target window within which the end point must lie after the touch probe has orbited the model. Enter half the side length of the target window for the side length.	**MP6390** **0.1** to **4.0000** [mm]

3-D touch probes and digitizing	
Radius measurement with the TT 130 touch probe: Probing direction	**MP6505.0 (traverse range 1) to 6505.2 (traverse range 3)** Positive probing direction in the angle reference axis (0° axis): **0** Positive probing direction in the +90° axis: **1** Negative probing direction in the angle reference axis (0° axis): **2** Negative probing direction in the +90° axis: **3**
Probing feed rate for second measurement with TT 120, stylus shape, corrections in TOOL.T	**MP6507** Calculate feed rate for second measurement with TT 130, with constant tolerance: **+0** Calculate feed rate for second measurement with TT 130, with variable tolerance: **+1** Constant feed rate for second measurement with TT 130: **+2**
Maximum permissible measuring error with TT 130 during measurement with rotating tool Required for calculating the probing feed rate in connection with MP6570	**MP6510** **0.001** to **0.999** [mm] (recommended input value: 0.005 mm)
Feed rate for probing a stationary tool with the TT 130	**MP6520** **1** to **3 000** [mm/min]
Radius measurement with the TT 130: Distance from lower edge of tool to upper edge of stylus	**MP6530.0 (traverse range 1) to MP6530.2 (traverse range 3)** **0.001** to **99.9999** [mm]
Set-up clearance in the tool axis above the stylus of the TT 130 for pre-positioning	**MP6540.0** **0.001** to **30 000.000** [mm]
Clearance zone in the machining plane around the stylus of the TT 130 for pre-positioning	**MP6540.1** **0.001** to **30 000.000** [mm]
Rapid traverse for TT 130 in the probe cycle	**MP6550** **10** to **10 000** [mm/min]
M function for spindle orientation when measuring individual teeth	**MP6560** **0** to **999**
Measuring rotating tools: Permissible rotational speed at the circumference of the milling tool Required for calculating rpm and probe feed rate	**MP6570** **1.000** to **120.000** [m/min]
Measuring rotating tools: Permissible rotational rpm	**MP6572** **0.000** to **1000.000** [rpm] If you enter 0, the speed is limited to 1000 rpm

3-D touch probes and digitizing	
Coordinates of the TT 120 stylus center relative to the machine datum	**MP6580.0 (traverse range 1)** X axis **MP6580.1 (traverse range 1)** Y axis **MP6580.2 (traverse range 1)** Z axis **MP6581.0 (traverse range 2)**, (not TNC 410) X axis **MP6581.1 (traverse range 2),** (not TNC 410) Y axis **MP6581.2 (traverse range 2),** (not TNC 410) Z axis **MP6582.0 (traverse range 3),** (not TNC 410) X axis **MP6582.1 (traverse range 3),** (not TNC 410) Y axis **MP6582.2 (traverse range 3),** (not TNC 410) Z axis

TNC displays, TNC editor	
Programming station	**MP7210** TNC with machine: **0** TNC as programming station with active PLC: **1** TNC as programming station with inactive PLC: **2**
Acknowledgment of POWER INTERRUPTED after switch-on	**MP7212** Acknowledge with key: **0** Acknowledge automatically: **1**
ISO programming: Set the block number increment	**MP7220** **0** to **150**
Disabling the selection of file types	**MP7224.0** All file types selectable via soft key: **+0** Disable selection of HEIDENHAIN programs (soft key SHOW .H): **+1** Disable selection of ISO programs (soft key SHOW .I): **+2** Disable selection of tool tables (soft key SHOW .T): **+4** Disable selection of datum tables (soft key SHOW .D): **+8** Disable selection of pallet tables (soft key SHOW .P): **+16** Disable selection of text files (soft key SHOW .A): **+32** (not TNC 410) Disable selection of point tables (soft key SHOW .PNT): **+64** (not TNC 410)

TNC displays, TNC editor	
Inhibit editing of particular file type (not TNC 410) **Note:** If a particular file type is inhibited, the TNC will erase all files of this type.	**MP7224.1** Do not disable editor: **+0** Disable editor for ■ HEIDENHAIN programs: **+1** ■ ISO programs: **+2** ■ Tool tables: **+4** ■ Datum tables: **+8** ■ Pallet tables: **+16** ■ Text files: **+32** ■ Point tables: **+64**
Configure pallet tables (not TNC 410)	**MP7226.0** Pallet table inactive: **0** Number of pallets per pallet table: **1** to **255**
Configure datum tables (not TNC 410)	**MP7226.1** Datum table inactive: **0** Number of datums per datum table: **1** to **255**
Program length as program control (not TNC 410)	**MP7229.0** Blocks **100** to **9 999**
Program length up to which FK blocks are permitted (not TNC 410)	**MP7229.1** Blocks **100** to **9 999**
Dialog language	**MP7230 on TNC 410** German: **0** English: **1** **MP7230 on TNC 426, TNC 430** English: **0** German: **1** Czech: **2** French: **3** Italian: **4** Spanish: **5** Portuguese: **6** Swedish: **7** Danish: **8** Finnish: **9** Dutch: **10** Polish: **11** Hungarian: **12** Reserved: **13** Russian: **14**
Set the internal clock of the TNC (not TNC 410)	**MP7235** Universal time (Greenwich time): **0** Central European Time (CET): **1** Central European Summertime: **2** Time difference to universal time: **-23** to **+23** [hours]

TNC displays, TNC editor	
Configure tool tables	**MP7260** Inactive: **0** Number of tools generated by the TNC when a new tool table is opened: **1** to **254** If you require more than 254 tools, you can expand the tool table with the function APPEND N LINES, see "Tool Data," page 99
Configure pocket tables	**MP7261.0 (magazine 1)** **MP7261.1 (magazine 2)** **MP7261.2 (magazine 3)** **MP7261.3 (magazine 4)** Inactive: **0** Number of pockets in the tool magazine: **1** to **254** If the value 0 is entered in MP7261.1 to MP7261.3, then only one tool magazine will be used.
Index tool numbers in order to be able to assign different compensation data to one tool number (not TNC 410)	**MP7262** Do not index: **0** Number of permissible indices: **1** to **9**
Soft key for pocket tables	**MP7263** Show the POCKET TABLE soft key in the tool table: **0** Do not show the POCKET TABLE soft key in the tool table: **1**

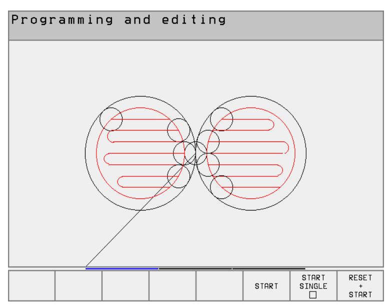
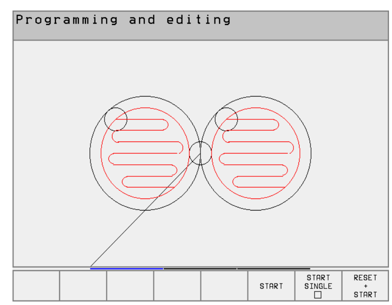
TNC displays, TNC editor	
Configure tool table (to omit from table, enter 0); Column number in the tool table	**MP7266.0** Tool name – NAME: **0** to **31**; column width: 16 characters **MP7266.1** Tool length – L: **0** to **31**; column width: 11 characters **MP7266.2** Tool radius – R: **0** to **31**; column width: 11 characters **MP7266.3** Tool radius 2 – R2: **0** to **31**; column width: 11 characters **MP7266.4** Oversize length – DL: **0** to **31**; column width: 8 characters **MP7266.5** Oversize radius – DR: **0** to **31**; column width: 8 characters **MP7266.6** Oversize radius 2 – DR2: **0** to **31**; column width: 8 characters **MP7266.7** Tool locked – TL: **0** to **31**; column width: 2 characters **MP7266.8** Replacement tool – RT: **0** to **31**; column width: 3 characters **MP7266.9** Maximum tool life – TIME1: **0** to **31**; column width: 5 characters **MP7266.10** Maximum tool life for TOOL CALL – TIME2: **0** to **31**; column width: 5 characters **MP7266.11** Current tool life – CUR. TIME: **0** to **31**; column width: 8 characters **MP7266.12** Tool comment – DOC: **0** to **31**; column width: 16 characters **MP7266.13** Number of teeth – CUT.: **0** to **31**; column width: 4 characters **MP7266.14** Tolerance for wear detection in tool length – LTOL: **0** to **31**; column width: 6 characters **MP7266.15** Tolerance for wear detection in tool radius – RTOL: **0** to **31**; column width: 6 characters **MP7266.16** Cutting direction – DIRECT.: **0** to **31**; column width: 7 characters **MP7266.17** PLC status – PLC: **0** to **31**; column width: 9 characters **MP7266.18** Offset of the tool in the tool axis in addition to MP6530 – TT:L-OFFS: **0** to **31** column width: 11 characters **MP7266.19** Offset of the tool between stylus center and tool center – TT:R-OFFS: **0** to **31** column width: 11 characters **MP7266.20** Tolerance for break detection in tool length – LBREAK: **0** to **31**; column width: 6 characters **MP7266.21** Tolerance for break detection in tool radius – RBREAK: **0** to **31**; column width: 6 characters **MP7266.22** Tooth length (Cycle 22) – LCUTS: **0** to **31**; column width: 11 characters **MP7266.23** Maximum plunge angle (Cycle 22) – ANGLE.: **0** to **31**; column width: 7 characters **MP7266.24** Tool type –TYP: **0** to **31**; column width: 5 characters **MP7266.25** Tool material – TMAT: **0** to **31**; column width: 16 characters **MP7266.26** Cutting data table – CDT: **0** to **31**; column width: 16 characters

TNC displays, TNC editor	
Configure tool table (to omit from table, enter 0); Column number in the tool table	**MP7266.27** PLC value – PLC-VAL: **0** to **31**; column width: 11 characters **MP7266.28** Center misalignment in reference axis – CAL-OFF1: **0** to **31**; column width: 11 characters **MP7266.29** Center misalignment in minor axis – CAL-OFF2: **0** to **31**; column width: 11 characters **MP7266.30** Spindle angle for calibration – CALL-ANG: **0** to **31**; column width: 11 characters
Configure pocket tables; Column number in the tool table (To omit from the table: enter 0)	**MP7267.0** Tool number – T: **0** to **7** **MP7267.1** Special tool – ST: **0** to **7** **MP7267.2** Fixed pocket – F: **0** to **7** **MP7267.3** Pocket locked – L: **0** to **7** **MP7267.4** PLC status – PLC: **0** to **7** **MP7267.5** Tool name from tool table – TNAME: **0** to **7** **MP7267.6** Comment from tool table – DOC: **0** to **7**
Manual Operation mode: Display of feed rate	**MP7270** Display feed rate F only if an axis direction button is pressed: **0** Display feed rate F even if no axis direction button is pressed (feed rate defined via soft key F or feed rate of the “slowest” axis): **1**
Decimal character	**MP7280** The decimal character is a comma: **0** The decimal character is a point: **1**
Display mode (not TNC 410)	**MP7281.0 Programming and Editing operating mode** **MP7281.1 Program Run operating modes** Always display multiple line blocks completely: **0** Display multiline blocks completely if the multiline block is the active block: **1** Display multiline blocks completely if the multiline block is being edited: **2**
Position display in the tool axis	**MP7285** Display is referenced to the tool datum: **0** Display in the tool axis is referenced to the tool face: **1**
Display step for the spindle position (not TNC 410)	**MP7289** 0.1 °: **0** 0.05 °: **1** 0.01 °: **2** 0.005 °: **3** 0.001 °: **4** 0.0005 °: **5** 0.0001 °: **6**

TNC displays, TNC editor	
Display step	**MP7290.0 (X axis) to MP7290.8 (9th axis, TNC 410 only to 4th axis)** 0.1 mm: **0** 0.05 mm: **1** 0.01 mm: **2** 0.005 mm: **3** 0.001 mm: **4** 0.0005 mm: **5** (not TNC 410) 0.0001 mm: **6** (not TNC 410)
Disable datum setting (not TNC 410)	**MP7295** Do not disable datum setting: **+0** Disable datum setting in the X axis: **+1** Disable datum setting in the Y axis: **+2** Disable datum setting in the Z axis: **+4** Disable datum setting in the IVth axis: **+8** Disable datum setting in the Vth axis: **+16** Disable datum setting in the 6th axis: **+32** Disable datum setting in the 7th axis: **+64** Disable datum setting in the 8th axis: **128** Disable datum setting in the 9th axis: **+256**
Disable datum setting with the orange axis keys	**MP7296** Do not disable datum setting: **0** Disable datum setting with the orange axis keys: **1**
Reset status display, Q parameters and tool data	**MP7300** Reset them all when a program is selected: **0** Reset them all when a program is selected and with M02, M30, END PGM: **1** Reset only status display and tool data when a program is selected: **2** Reset only status display and tool data when a program is selected and with M02, M30, END PGM: **3** Reset status display and Q parameters when a program is selected: **4** Reset status display and Q parameters when a program is selected and with M02, M30, END PGM: **5** Reset status display when a program is selected: **6** Reset status display when a program is selected and with M02, M30, END PGM: **7**
Graphic display mode	**MP7310** Projection in three planes according to ISO 6433, projection method 1: **+1** Projection in three planes according to ISO 6433, projection method 2: **+1** Do not rotate coordinate for graphic display: **+0** Rotate coordinate system for graphic display by 90°: **+2** Display new BLK FORM in Cycle 7 DATUM SHIFT referenced to the old datum: **+0** Display new BLK FORM in Cycle 7 DATUM SHIFT referenced to the new datum: **+4** Do not show cursor position during projection in three planes: **+0** Show cursor position during projection in three planes: **+8**
Settings for the programming graphics (not TNC 426, TNC 430)	**MP7311** Do not show penetration points as circle: **+0** Show penetration points as circle: **+1** Do not show meander paths in cycles: **+0** Show meander paths in cycles: **+2** Do not show compensated paths: **+0** Show compensated paths: **+4**

TNC displays, TNC editor	
Graphic simulation without programmed tool axis: Tool radius (not TNC 410)	**MP7315** **0** to **99 999.9999** [mm]
Graphic simulation without programmed tool axis: Penetration depth (not TNC 410)	**MP7316** **0** to **99 999.9999** [mm]
Graphic simulation without programmed tool axis: M function for start (not TNC 410)	**MP7317.0** **0** to **88** (0: Function inactive)
Graphic simulation without programmed tool axis: M function for end (not TNC 410)	**MP7317.1** **0** to **88** (0: Function inactive)
Set the screen saver (not TNC 410) Enter the time after which the TNC should start the screen saver	**MP7392** **0** to **99** [min] (0: Function inactive)

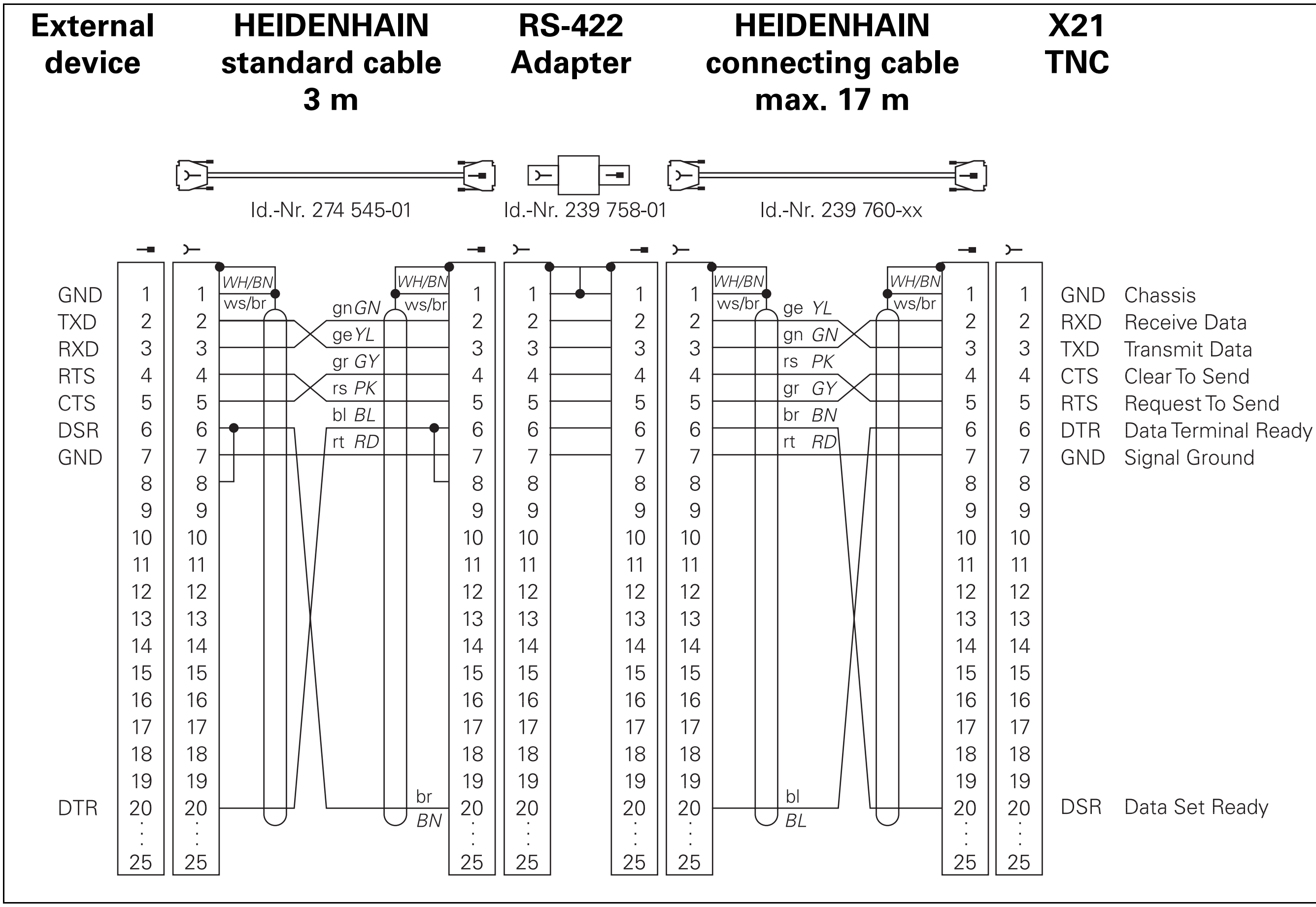
Machining and program run	
Cycle 17: Oriented spindle stop at beginning of cycle	**MP7160** Oriented spindle stop: **0** No oriented spindle stop: **1**
Effect of Cycle 11 SCALING FACTOR	**MP7410** SCALING FACTOR effective in 3 axes: **0** SCALING FACTOR effective in the working plane only: **1**
Manage tool data/calibration data	**MP7411** Overwrite current tool data by the calibrated data from the 3-D touch probe system: **+0** Current tool data are retained: **+1** Manage calibrated data in the calibration menu: **+0** (not TNC 410) Manage calibrated data in the tool table: **+2** (not TNC 410)

Machining and program run	
SL Cycles	**MP7420** Mill channel around the contour - clockwise for islands and counterclockwise for pockets: **+0** Mill channel around the contour - clockwise for pockets and counterclockwise for islands: **+1** First mill the channel, then rough out the contour: **+0** First rough out the contour, then mill the channel: **+2** Combine compensated contours: **+0** Combine uncompensated contours: **+4** Complete one process for all infeeds before switching to the other process: **+0** Mill channel and rough-out for each infeed depth before continuing to the next depth: **+8** The following note applies to the Cycles G56, G57, G58, G59, G121, G122, G123 and G124: At the end of the cycle, move the tool to the position that was last programmed before the cycle call: **+0** At the end of the cycle, retract the tool in the tool axis only: **+16**
SL cycles, Group I, operating procedure (not TNC 426, TNC 430)	**MP7420.1** Rough-out separate areas together, plunging with every pass: **+0** Rough-out separate areas separately, plunging only once for each area: **+1** Bit 1 to bit 7: reserved   **MP7420.1 = 0** (small circles = penetration) **MP7420.1 = 1**
Cycle 4 POCKET MILLING and Cycle 5 CIRCULAR POCKET MILLING: Overlap factor	**MP7430** **0.1** to **1.414**
Permissible deviation of circle radius between circle end point and circle starting point (not TNC 410)	**MP7431** **0.0001** to **0.016** [mm]

Machining and program run	
Operation of various miscellaneous functions M **Note:** The k_V factors for position loop gain are set by the machine tool builder. Refer to your machine manual.	**MP7440** Program stop with M06: **+0** No program stop with M06: **+1** No cycle call with M89: **+0** Cycle call with M89: **+2** Program stop with M functions: **+0** No program stop with M functions: **+4** k_V factors cannot be switched through M105 and M106: **+0 (not TNC 410)** k_V factors can be switched through M105 and M106: **+8 (not TNC 410)** Reduce the feed rate in the tool axis with M103 F.. Function inactive: **+0** Reduce the feed rate in the tool axis with M103 F.. Function active: **+16** Exact stop for positioning with rotary axes not active: **+0 (not TNC 410)** Exact stop for positioning with rotary axes active: **+32 (not TNC 410)**
Error message during cycle call (not TNC 410)	**MP7441** Error message when M3/M4 not active: **0** Suppress error message when M3/M4 not active: **+1** Reserved: **+2** Suppress error message when positive depth programmed: **+0** Output error message when negative depth programmed: **+4**
M function for spindle orientation in the fixed cycles	**MP7442** Function inactive: **0** Orientation directly through the NC: **-1** M function for orienting the spindle: **1 to 999**
Maximum contouring speed at feed rate override setting of 100% in the Program Run modes	**MP7470** **0** to **99 999** [mm/min]
Feed rate for rotary-axis compensation movements (not TNC 410)	**MP7471** **0** to **99 999** [mm/min]
Datums from a datum table are referenced to the	**MP7475** Workpiece datum: **0** Machine datum: **1**
Running pallet tables (not TNC 410)	**MP7683** Program Run, Single Block: Run one line of the active NC program at every NC start; Program Run, Full Sequence: Run the entire NC program at every NC start: **+0** Program Run, Single Block: Run the entire NC program at every NC start:**+1** Program Run, Full Sequence: Run all NC programs up to the next pallet at every NC start: **+2** Program Run, Full Sequence: Run the entire NC pallet file at every NC start:**+4** Program Run, Full Sequence: If running of the complete pallet file is selected (+4), then run the pallet file without interruption, i.e. until you press NC stop: **+8**

13.2 Pin Layout and Connecting Cable for the Data Interfaces

RS-232-C/V.24 Interface HEIDEHAIN devices

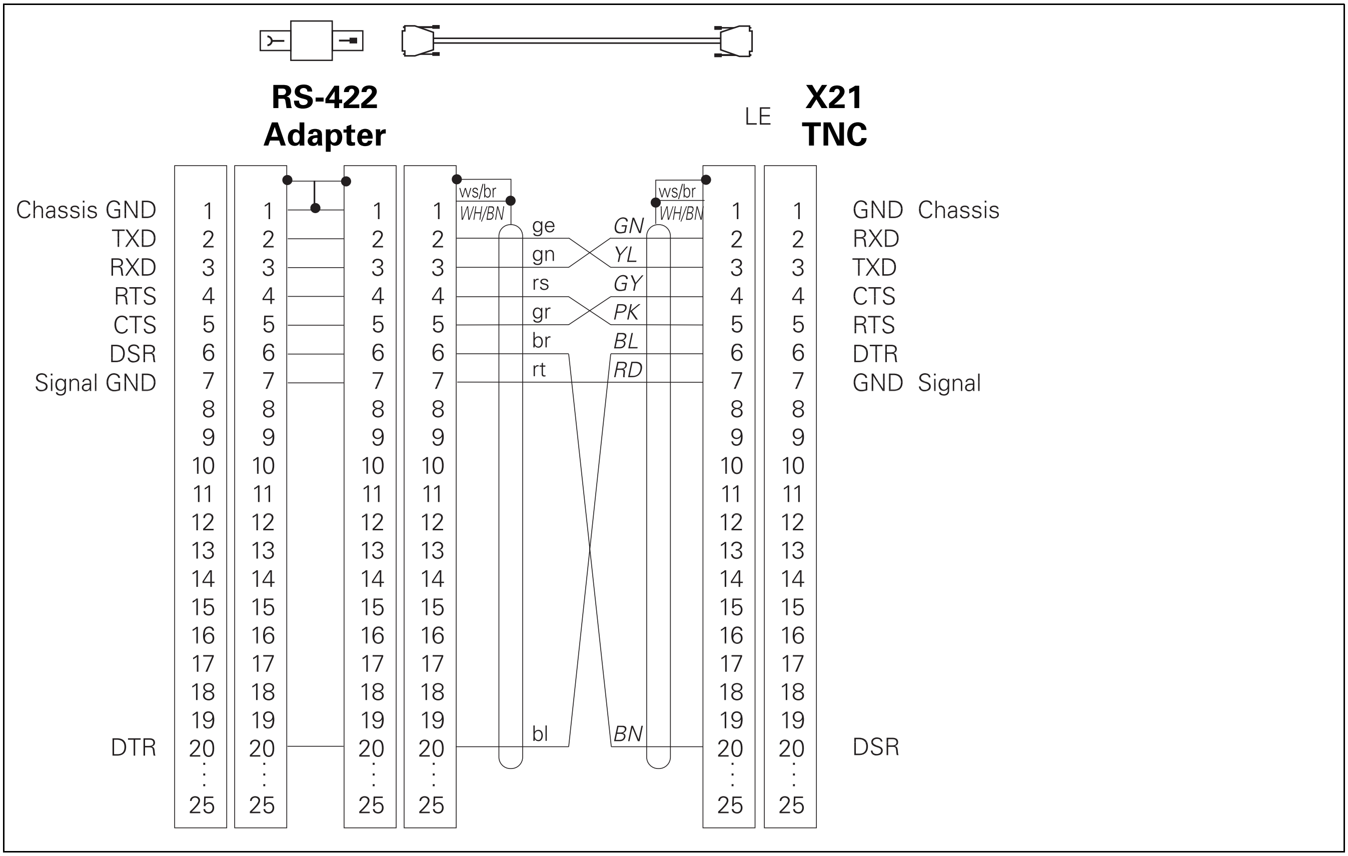


The connector pin layout on the adapter block differs from that on the TNC logic unit (X21).

Non-HEIDENHAIN devices

The connector pin layout of a non-HEIDENHAIN device may differ considerably from that on a HEIDENHAIN device.

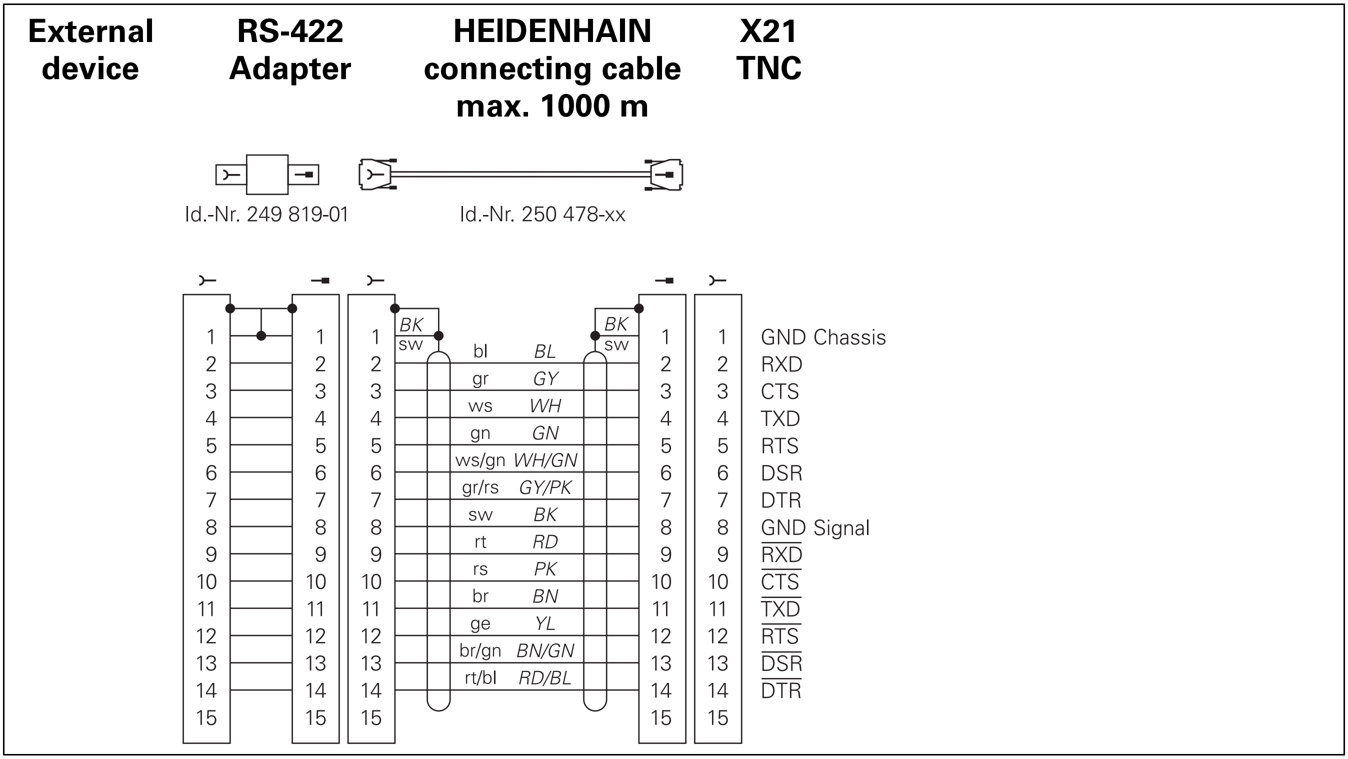
This often depends on the unit and type of data transfer. The figure below shows the connector pin layout on the adapter block.



RS-422/V.11 interface (not TNC 410)

Only non-HEIDENHAIN devices are connected to the RS-422 interface.

> The pin layouts on the TNC logic unit (X22) and on the adapter block are identical.



Ethernet interface RJ45 socket (option, not TNC 410)

Maximum cable length:Unshielded:100 m
Shielded:400 m

Pin	Signal	Description
1	TX+	Transmit Data
2	TX–	Transmit Data
3	REC+	Receive Data
4	Vacant	
5	Vacant	
6	REC–	Receive Data
7	Vacant	
8	Vacant	

Ethernet interface BNC socket (option, not TNC 410)

Maximum cable length:180 m

Pin	Signal	Description
1	Data (RXI, TXO)	Inner conductor (core)
2	GND	Shielding

13.3 Technical Information

TNC features

TNC features	
Description	Contouring control for machines with up to 9 axes (TNC 410: up to 4 axes) plus oriented spindle stop; TNC 410, TNC 426 CB and TNC 430 CA feature analog speed control, the TNC 410 PA, TNC 426 PB and TNC 430 PB feature digital speed control and integrated current controller.
Components	■ Logic unit ■ Keyboard ■ Color visual display unit with soft keys
Data interfaces	■ RS-232-C / V.24 ■ RS-422/V.11 interface (not TNC 410) ■ Ethernet interface (option, not TNC 410) ■ Expanded data interface with LSV-2 protocol for remote operation of the TNC through the data interface with the HEIDENHAIN software TNCremo
Simultaneous axis control for contour elements	■ Straight lines up to 5 axes (TNC 410: up to 3 axes) Export versions TNC 426 CF, TNC 426 PF, TNC 430 CE, TNC 430 PE: 4 axes ■ Circles: up to 3 axes (with tilted working plane), TNC 410: 2 axes ■ Helices: 3 axes
Look-ahead	■ Defined rounding of discontinuous contour transitions (such as for 3-D surfaces) ■ Collision prevention with the SL cycle for open contours ■ Geometry precalculation of radius-compensated positions for feed rate adaptation with M120
Background programming	One part program can be edited while the TNC runs another program
Graphics	■ Interactive Programming graphics ■ Test Run graphics ■ Program run graphics (not TNC 410)
File types	■ HEIDENHAIN conversational programming ■ ISO programs ■ Tool tables ■ Cutting data tables (not TNC 410) ■ Datum tables ■ Point tables ■ Pallet files (not TNC 410) ■ Text files ■ System files (not TNC 410)

TNC features	
Program memory	■ Hard disk with 1500 MB for NC programs (TNC 410: 256 KB, i.e. approx. 10 000 NC blocks with battery buffer backup) ■ Any number of files (TNC 410: up to 64 files)
Tool definitions	Up to 254 tools in program, any number of tools in tables (TNC 410: up to 254)
Programming support	■ Functions for approaching and departing the contour ■ Integrated pocket calculator (not TNC 410) ■ Structuring long programs (not TNC 410) ■ Comment blocks ■ Direct help on output error messages (context-sensitive, not TNC 410)) ■ Help function for ISO programming (not TNC 426, TNC 430)

Programmable functions	
Contour elements	■ Straight line ■ Chamfer ■ Circular path ■ Circle center ■ Circle radius ■ Tangentially connecting circle ■ Corner rounding ■ Straight lines and circular arcs for contour approach and departure ■ B spline (only conversational dialog, not TNC 410)
FK free contour programming	For all contour elements not dimensioned for conventional NC programming
Three-dimensional tool radius compensation	For changing tool data without having to recalculate the program
Program jumps	■ Subprogram ■ Program section repeat ■ Program as subprogram
Fixed cycles	■ Drilling cycles for drilling, pecking, reaming, boring, tapping with a floating tap holder, rigid tapping ■ Cycles for milling internal and external threads (not TNC 410) ■ Milling and finishing rectangular and circular pockets ■ Cycles for multipass milling of flat and twisted surfaces ■ Cycles for milling linear and circular slots ■ Linear and circular hole patterns ■ Milling pockets and islands from a list of subcontour elements ■ Interpolation of cylinder surface (not TNC 410)

Programmable functions	
Coordinate transformations	■ Datum shift ■ Mirror image ■ Rotation ■ Scaling factor ■ Tilting the Working Plane (not TNC 410)
3-D touch probe applications	■ Touch probe functions for compensating workpiece misalignment ■ Touch probe functions for setting datums ■ Touch probe functions for automatic workpiece measurement ■ Digitizing 3-D surfaces with the measuring touch probe (optional, not TNC 410) ■ Digitizing 3-D surfaces with the triggering touch probe (optional) ■ Automatic tool measurement with the TT 130 (TNC 410: only conversational dialog)
Mathematical functions	■ Basic arithmetic +, –, x and / ■ Trigonometry sin, cos, tan, arc sin, arc cos, arc tan ■ Square root and root sum of squares ■ Squaring (SQ) ■ Powers (^) ■ Constant PI (3.14) ■ Logarithms ■ Exponential function ■ Negation (NEG) ■ Forming an integer (INT) ■ Forming an absolute number (ABS) ■ Truncating values before the decimal point (FRAC) ■ Functions for calculating circles ■ Logical comparisons (greater than, less than, equal to, not equal to)

TNC Specifications	
Block processing time	4 ms/block, 6 ms/block, 20 ms/block for blockwise execution via data interface
Control loop cycle time	■ TNC 410 Contouring interpolation: 6 ms ■ TNC 426 PB, TNC 430 PA: Contouring interpolation: 3 ms Fine interpolation: 0.6 ms (speed) ■ TNC 426 CB, TNC 430 CA: Contouring interpolation: 3 ms Fine interpolation: 0.6 ms (contour) ■ TNC 426 M, TNC 430 M: Contouring interpolation: 3 ms Fine interpolation: 0.6 ms (speed)

TNC Specifications	
Data transfer rate	Maximum 115 200 baud via V.24/V.11 Maximum 1 Mbaud via Ethernet interface (optional, not TNC 410)
Ambient temperature	■ Operation: 0° C to +45° C (32° to 113° F) ■ Storage: –30°C to +70°C (–22° F to 158° F)
Traverse range	Maximum 100 m (3973 inches) TNC 410: Maximum 30 m (1181 inches)
Traversing speed	Maximum 300 m/min (11 811 ipm) TNC 410: Maximum 100 m/min (3 937 ipm)
Spindle speed	Maximum 99 999 rpm
Input range	■ Minimum 0.1µm (0.00001 inches) or 0.0001° (TNC 410: 1 µm) ■ Maximum 99 999.999 mm (3937 in.) or 99 999.999° TNC 410: Maximum 30 000 mm (1181 inches) or 30 000.000°

Input format and unit of TNC functions	
Positions, coordinates, circle radii, chamfer lengths	-99 999.9999 to +99 999.9999 (5.4: places before decimal point, places after decimal point) [mm]
Tool numbers	0 to 32 767.9 (5.1)
Tool names	16 characters, enclosed by quotation marks with TOOL CALL. Permitted special characters: #, $, %, &, -
Delta values for tool compensation	-99.9999 to +99.9999 (2.4) [mm]
Spindle speeds	0 to 99 999.999 (5.3) [rpm]
Feed rates	0 to 99 999.999 (5.3) [mm/min] or [mm/rev]
Dwell time in Cycle 04	0 to 3 600.000 (4.3) [s]
Thread pitch in various cycles	-99.9999 to +99.9999 (2.4) [mm]
Angle of spindle orientation	0 to 360.0000 (3.4) [°]
Angle for polar coordinates, rotation, tilting the working plane	-360.0000 to 360.0000 (3.4) [°]
Polar coordinate angle for helical interpolation (G12/G13)	-5 400.0000 to 5 400.0000 (4.4) [°]
Datum numbers in Cycle 7	0 to 2 999 (4.0)
Scaling factor in Cycles 11 and 26	0.000 001 to 99.999 999 (2.6)
Miscellaneous functions M	0 to 999 (1.0)
Q parameter numbers	0 to 399 (1.0)
Q parameter values	-99 999.9999 to +99 999.9999 (5.4)
Labels (G98) for program jumps	0 to 254 (3.0)

Input format and unit of TNC functions	
Number of program section repeats L	1 to 65 534 (5.0)
Error number with Q parameter function D14	0 to 1 099 (4.0)

13.4 Exchanging the Buffer Battery

A buffer battery supplies the TNC with current to prevent the data in RAM memory from being lost when the TNC is switched off.

If the TNC displays the error message **Exchange buffer battery**, then you must replace the batteries:

> To exchange the buffer battery, first switch off the TNC!
>
> The buffer battery must be exchanged only by trained service personnel!

TNC 410 CA/PA, TNC 426 CB/PB, TNC 430 CA/PA

Battery type:Three AA-size cells, leak-proof, IEC designation “LR6”

1. Open the logic unit: The buffer batteries are located next to the power supply unit.
2. Open the battery compartment: With a screwdriver, open the cover by turning it counterclockwise by 90°.
3. Exchange the batteries and take care to properly close the battery compartment again.

TNC 410 M, TNC 426 M, TNC 430 M

Battery type:1 Lithium battery, type CR 2450N (Renata) ID No. 315 878-01

1. Open the logic unit: The buffer battery is located to the right of the EPROMs of the NC software
2. Exchange the battery. The new battery can only be inserted the right way around.

13.5 Addresses (ISO)

G functions

Group	G	Function	Blockwise function	Note
Positioning	00	Straight-line interpolation, Cartesian coordinates, rapid traverse		page 127
	01	Straight-line interpolation, Cartesian coordinates		page 127
	02	Circular interpolation, Cartesian coordinates, clockwise	■ (with R)	page 131
	03	Circular interpolation, Cartesian coordinates, counterclockwise	■ (with R)	page 131
	05	Circular interpolation, Cartesian coordinates, without indication of direction		page 131 page 134
	06	Circular interpolation, Cartesian coordinates, tangential contour approach		
	07	Paraxial positioning block	■	
	10	Straight-line interpolation, polar coordinates, rapid traverse		page 140
	11	Straight-line interpolation, polar coordinates		page 140
	12	Circular interpolation, polar coordinates, clockwise		page 140
	13	Circular interpolation, polar coordinates, counterclockwise		page 140
	15	Circular interpolation, polar coordinates, without indication of direction		page 140
	16	Circular interpolation, polar coordinates, tangential contour approach		page 141
Machining contours, approaching/departing	24	Chamfer with length R		page 128
	25	Corner rounding with radius R		page 129
	26	Tangential approach of a contour with R		page 124
	27	Tangential departure of a contour with R		page 124
Cycles for drilling, tapping and thread milling	83	Pecking		page 185
	84	Tapping with a floating tap holder		page 199
	85	Rigid tapping		page 202
	86	Thread cutting (not TNC 410)		page 205
	200	Drilling		page 186
	201	Reaming		page 187
	202	Boring		page 189
	203	Universal drilling		page 191
	204	Back boring		page 193
	205	Universal pecking (not TC 410)		page 195
	206	Tapping with a floating tap holder (not TNC 410)		page 200
	207	Rigid tapping (not TNC 410)		page 203
	208	Bore milling (not TNC 410)		page 197
	209	Tapping with chip breaking (not TNC 410)		page 206
	262	Thread milling (not TNC 410)		page 210
	263	Thread milling/countersinking (not TNC 410)		page 212
	264	Thread drilling/milling (not TNC 410)		page 216
	265	Helical thread drilling/milling (not TNC 410)		page 220
	267	Outside thread milling (not TNC 410)		page 223

Group	G	Function	Blockwise function	Note
Cycles for milling pockets, studs and slots	74	Slot milling		page 244
	75	Rectangular pocket milling in clockwise direction		page 232
	76	Circular path in counterclockwise direction		page 232
	77	Circular pocket milling in clockwise direction		page 238
	78	Circular pocket milling in counterclockwise direction		page 238
	210	Slot milling with reciprocating plunge		page 246
	211	Round slot with reciprocating plunge		page 248
	212	Rectangular pocket finishing		page 234
	213	Rectangular stud finishing		page 236
	214	Circular pocket finishing		page 240
	215	Circular stud finishing		page 242
Cycles for creating point patterns	220	Circular pattern		page 254
	221	Linear pattern		page 256
Cycles for creating complex contours	37	Definition of pocket contour		page 261
	56	Pilot drilling of the contour pocket (with G37) SLI		page 262
	57	Rough-out of the contour pocket (with G37) SLI		page 263
	58	Contour milling in clockwise direction (with G37) SLI		page 264
	59	Contour milling in counterclockwise direction (with G37) SLI		page 264
	37	Definition of pocket contour		page 265
	120	Contour data (not TNC 410)		page 270
	121	Pilot drilling (with G37) SLII (not TNC 410)		page 271
	122	Rough-out (with G37) SLII (not TNC 410)		page 272
	123	Floor finishing (with G37) SLII (not TNC 410)		page 273
	124	Side finishing (with G37) SLII (not TNC 410)		page 274
	125	Contour train (with G37, not TNC 410)		page 275
	127	Cylinder surface (with G37, not TNC 410)		page 277
	128	Cylindrical surface slot (with G37, not TNC 410)		page 279
Cycles for multipass milling	60	Running point tables (not TNC 410)		page 288
	230	Multipass milling of plane surfaces		page 289
	231	Multipass milling of tilted surfaces		page 291
Coordinate transformation cycles	28	Mirror image		page 300
	53	Datum shift in a datum table		page 296
	54	Datum shift in program		page 295
	72	Scaling factor		page 303
	73	Rotation of the coordinate system		page 302
	80	Machining plane (not TNC 410)		page 304
Special cycles	04	Dwell time	■	page 311
	36	Oriented spindle stop		page 312
	39	Cycle for program call, program call via G79	■	page 311
	62	Tolerance deviation for fast contour milling (not TNC 410)		page 313
Cycles for measuring workpiece misalignment (not TNC 410)	400	Basic rotation from two points	■	See User's Manual "Touch Probe Cycles"
	401	Basic rotation from two holes	■	
	402	Basic rotation from two studs	■	
	403	Compensating misalignment with rotary axis	■	
	404	Setting a basic rotation directly	■	
	405	Compensating misalignment with the C axis	■	

Group	G	Function	Blockwise function	Note
Cycles for automatic datum setting (not TNC 410)	410 411 412 413 414 415 416 417 418	Datum in center of rectangular pocket Datum in center of rectangular stud Datum in center of circular pocket/hole Datum in center of circular stud Datum in inside corner Datum in outside corner Datum in center of bolt hole circle Datum in the touch probe axis Datum in intersection of two connecting lines each connecting two holes	■ ■ ■ ■ ■ ■ ■ ■ ■	See User's Manual "Touch Probe Cycles"
Cycles for automatic workpiece measurement (not TNC 410)	55 420 421 422 423 424 425 426 427 430 431	Measuring any coordinate in any axis Measuring angles Measuring position and diameter of a circular pocket/hole Measuring position and diameter of a circular stud Measuring position and diameter of a rectangular pocket Measuring position and diameter of a rectangular stud Measuring the slot width Measuring a ridge Measuring any coordinate in any axis Measuring position and diameter of a bolt hole circle Measuring a plane	■ ■ ■ ■ ■ ■ ■ ■ ■ ■ ■	See User's Manual "Touch Probe Cycles"
Cycles for automatic tool measurement (not TNC 410)	480 481 482 483	Calibrating the TT Measuring tool length Measuring tool radius Measuring tool length and radius	■ ■ ■ ■	See User's Manual "Touch Probe Cycles"
Cycles in general	79	Call the cycle	■	page 177
Selection of the machining plane	17 18 19 20	Plane selection XY, tool axis Z Plane selection ZX, tool axis Y Plane selection YZ, tool axis X Tool axis IV		page 109
Capture of coordinates	29	Transfer the last nominal position value as a pole		page 130
Define the workpiece blank	30 31	Define workpiece blank for graphics, min. point Define workpiece blank for graphics, max. point		page 71
Influencing the program run	38	Program run STOP		
	40 41 42 43 44	No tool compensation (R0) Tool radius compensation, to the left of the contour (RL) Tool radius compensation, to the right of the contour (RR) Paraxial compensation, lengthening (R+) Paraxial compensation, shortening (R–)		page 113
Tools	51 99	Next tool number (in active central tool memory) Tool definition	■ ■	page 110 page 100

Group	G	Function	Blockwise function	Note
Unit of measure	70 71	Unit of measure: inches (set at start of program) Unit of measure: millimeters (set at start of program)		page 72
Dimensions	90 91	Absolute dimensions Incremental dimensions		page 41 page 41
Subprograms	98	Setting a label number	■	

Assigned addresses

Designation	Function
%	Program start or program call
#	Datum number with Cycle G53
A **B** **C**	Rotation about X axis Rotation about Y axis Rotation about Z axis
D	Definition of parameters (program parameters Q)
DL **DR**	Length wear compensation with tool call Radius wear compensation with tool call
E	Tolerance for M112 and M124
F **F** **F** **F**	Feed rate Dwell time with G04 Scaling factor with G72 Factor for feed-rate reduction with M103
G	Preparatory function, cycle definition
H **H** **H**	Polar coordinates angle in incremental value/absolute value Rotation angle with G73 Tolerance angle for M112
I **J** **K**	Z coordinate of the circle center/pole Y coordinate of the circle center/pole Z coordinate of the circle center/pole
L **L** **L**	Setting a label number with G98 Jump to a label number Tool length with G99
LA	Number of blocks for block scan with M120
M	Miscellaneous Functions
N	Block number
P **P**	Cycle parameters in machining cycles Parameters in parameter definitions

Designation	Function
Q	Program parameters/Cycle parameters
R **R** **R** **R** **R**	Polar coordinate radius Circular radius with G02/G03/G05 Rounding radius with G25/G26/G27 Chamfer section with G24 Tool radius with G99
S **S**	Spindle speed Oriented spindle stop with G36
T **T**	Tool definition with G99 Tool call
U **V** **W**	Linear movement parallel to X axis Linear movement parallel to Y axis Linear movement parallel to Z axis
X **Y** **Z**	X axis Y axis Z axis
*****	End of block

Parameter functions

Parameter definition	Function	Note
D00	Assign	page 333
D01 **D02** **D03** **D04**	Addition Subtraction Multiplication Division	page 333 page 333 page 333 page 333
D05	Root	page 333
D06 **D07**	Sine Cosine	page 336 page 336
D08	Root sum of squares	page 336
D09 **D10** **D11** **D12**	If equal, go to If not equal, go to If greater than, go to If less than, go to	page 338 page 338 page 338 page 338
D13	Angle from c ¦ sin a and c ¦ cos a)	page 336
D14	Error number	page 341
D15	Print	page 345
D19	Transfer of values to the PLC	page 346

Q

R

S

S

T

T

U

V

W

Table of Miscellaneous Functions

M	Effect	Effective at block start	Effective at block end	Page
M00	Stop program/Spindle STOP/Coolant OFF		■	page 149
M01	Optional program STOP		■	page 386
M02	Stop program/Spindle STOP/Coolant OFF/Clear status display (depending on machine parameter)/Go to block 1		■	page 149
M03	Spindle ON clockwise	■		page 149
M04	Spindle ON counterclockwise	■		
M05	Spindle STOP		■	
M06	Tool change/Stop program run (depending on machine parameter)/Spindle STOP		■	page 149
M08	Coolant ON	■		page 149
M09	Coolant OFF		■	
M13	Spindle ON clockwise/Coolant ON	■		page 149
M14	Spindle ON counterclockwise/Coolant ON	■		
M30	Same function as M02		■	page 149
M89	Vacant miscellaneous function **or**	■		page 177
	Cycle call, modally effective (depending on machine parameter)		■	
M90	Only in lag mode: Constant contouring speed at corners		■	page 153
M91	Within the positioning block: Coordinates are referenced to machine datum	■		page 150
M92	Within the positioning block: Coordinates are referenced to position defined by machine tool builder, such as tool change position	■		page 150
M94	Reduce display of rotary axis to value under 360°	■		page 166
M97	Machine small contour steps		■	page 157
M98	Machine open contours completely		■	page 158
M99	Blockwise cycle call		■	page 177

M	Effect	Effective at block block start	end	Page
M101	Automatic tool change with replacement tool if maximum tool life has expired	■		page 110
M102	Reset M101		■	
M103	Reduce feed rate during plunging to factor F (percentage)	■		page 158
M107	Suppress error message for replacement tools	■		page 110
M108	Reset M107		■	
M109	Constant contouring speed at tool cutting edge (increase and decrease feed rate)	■		page 160
M110	Constant contouring speed at tool cutting edge (feed rate decrease only)	■		
M111	Reset M109/M110		■	
M112	Entering contour transitions between contour elements	■		page 154
M113	Reset M112 (not TNC 426, TNC 430)			
M120	Pre-calculate radius-compensated contour (LOOK AHEAD)	■		page 160
M124	Contour filter (not TNC 426, TNC 430)	■		page 156
M126	Shorter-path traverse of rotary axes	■		page 165
M127	Reset M126		■	

Additional M functions for TNC 426, TNC 430

M	Effect	Effective at block block start	end	Page
M104	Reactivate the datum as last defined	■		page 152
M105	Machining with second kv factor	■		page 435
M106	Machining with first kv factor	■		
M114	Automatic compensation of machine geometry when working with tilted axes	■		page 167
M115	Reset M114		■	
M116	Feed rate for angular axes in mm/min	■		page 164
M117	Reset M116		■	
M118	Superimpose handwheel positioning during program run	■		page 162
M128	Maintain the position of the tool tip when positioning with tilted axes (TCPM)	■		page 168
M129	Reset M128		■	
M130	Moving to position in an untilted coordinate system with a tilted working plane	■		page 152
M134	Exact stop at nontangential contour transitions when positioning with rotary axes	■		page 169
M135	Reset M134		■	
M136	Feed rate F in millimeters per spindle revolution	■		page 159
M137	Reset M136		■	
M138	Select tilting axes	■		page 170
M142	Delete modal program information	■		page 163

M	Effect	Effective at block	block start	end	Page
M143	Delete basic rotation		■		page 163
M144 M145	Compensating the machine's kinematic configuration for ACTUAL/NOMINAL positions at end of block Reset M144		■	 ■	page 171
M200 M201 M202 M203 M204	Laser cutting: Output programmed voltage directly Laser cutting: Output voltage as a function of distance Laser cutting: Output voltage as a function of speed Laser cutting: Output voltage as a function of time (ramp) Laser cutting: Output voltage as a function of time (pulse)		■ ■ ■ ■ ■		page 172

ISO Function Overview

TNC 410, TNC 426, TNC 430

M functions	
M00	Stop program/Spindle STOP/Coolant OFF
M01	Optional program STOP
M02	Stop program run/Spindle STOP/Coolant OFF/Clear status display (depending on machine parameter)/Go to block 1
M03	Spindle ON clockwise
M04	Spindle ON counterclockwise
M05	Spindle STOP
M06	Tool change/Stop program run (depending on machine parameter)/Spindle STOP
M08	Coolant ON
M09	Coolant OFF
M13	Spindle ON clockwise/Coolant ON
M14	Spindle ON counterclockwise/Coolant ON
M30	Same function as M02
M89	Vacant miscellaneous function or Cycle call, modally effective (depending on machine parameter)
M90	Only in lag mode: Constant contouring speed at corners
M99	Blockwise cycle call
M91	Within the positioning block: Coordinates are referenced to machine datum
M92	Within the positioning block: Coordinates are referenced to position defined by machine tool builder, such as tool change position
M94	Reduce display of rotary axis to value under 360°
M97	Machine small contour steps
M98	Machine open contours completely
M101	Automatic tool change with replacement tool if maximum tool life has expired
M102	Reset M101
M103	Reduce feed rate during plunging to factor F (percentage)
M107	Suppress error message for replacement tools
M108	Reset M107
M109	Constant contouring speed at tool cutting edge (increase and decrease feed rate)
M110	Constant contouring speed at tool cutting edge (feed rate decrease only)
M111	Reset M109/M110

M functions	
M112	Entering contour transitions between two contour elements (not TNC 426, TNC 430)
M113	Cancel M112
M120	Pre-calculate radius-compensated contour (LOOK AHEAD)
M124	Contour filter (not TNC 426, TNC 430)
M126	Shorter-path traverse of rotary axes
M127	Reset M126

Additional M functions for TNC 426, TNC 430

M functions	
M104	Reactivate the datum as last defined
M105	Machining with second kv factor
M106	Machining with first kv factor
M114	Automatic compensation of machine geometry when working with tilted axes:
M115	Reset M114
M116	Feed rate for angular axes in mm/min
M117	Reset M116
M118	Superimpose handwheel positioning during program run
M128	Maintain the position of the tool tip when positioning with tilted axes (TCPM)
M129	Reset M128
M130	Moving to position in an untilted coordinate system with a tilted working plane
M134	Exact stop at nontangential contour transitions when positioning with rotary axes
M135	Reset M134
M136	Feed rate F in millimeters per spindle revolution
M137	Reset M136
M138	Select tilting axes
M142	Delete modal program information
M143	Delete basic rotation
M144	Compensating the machine's kinematic configuration for ACTUAL/NOMINAL positions at end of block
M145	Reset M144
M200	Laser cutting: Output programmed voltage directly
M201	Laser cutting: Output voltage as a function of distance
M202	Laser cutting: Output voltage as a function of speed
M203	Laser cutting: Output voltage as a function of time (ramp)
M204	Laser cutting: Output voltage as a function of time (pulse)

G functions

Tool Movements

G00	Straight-line interpolation, Cartesian coordinates, rapid traverse
G01	Straight-line interpolation, Cartesian coordinates
G02	Circular interpolation, Cartesian coordinates,
G03	clockwise Circular interpolation, Cartesian coordinates, counterclockwise
G05	Circular interpolation, Cartesian coordinates, without indication of direction
G06	Circular interpolation, Cartesian coordinates, tangential contour connection
G07*	Paraxial positioning block
G10	Straight-line interpolation, polar coordinates, rapid traverse
G11	Straight-line interpolation, polar coordinates
G12	Circular interpolation, polar coordinates, clockwise
G13	Circular interpolation, polar coordinates, counterclockwise
G15	Circular interpolation, polar coordinates, without indication of direction
G16	Circular interpolation, polar coordinates, tangential contour connection

Chamfer/Rounding/Approach contour/Depart contour

G24*	Chamfer with length R
G25*	Corner rounding with radius R
G26*	Tangential contour approach with tool radius R
G27*	Tangential contour departure with tool radius R

Define the tool

G99*	With tool number T, length L, radius R

Tool radius compensation

G40	No tool radius compensation
G41	Tool radius compensation, left of the contour
G42	Tool radius compensation, right of the contour
G43	Paraxial compensation for G07, lengthening
G44	Paraxial compensation for G07, shortening

Blank form definition for graphics

G30	(G17/G18/G19) min. point
G31	(G90/G91) max. point

Cycles for drilling, tapping and thread milling

G83	Pecking
G84	Tapping with a floating tap holder
G85	Rigid tapping
G86	Thread cutting (not TNC 410)
G200	Drilling
G201	Reaming
G202	Boring
G203	Universal drilling
G204	Back boring
G205	Universal pecking (not TNC 410)
G206	Tapping with a floating tap holder (not TNC 410)
G207	Rigid tapping (not TNC 410)
G208	Bore milling (not TNC 410)
G209	Tapping with chip breaking (not TNC 410)

G functions

Cycles for drilling, tapping and thread milling

G262	Thread milling (not TNC 410)
G263	Thread milling/countersinking (not TNC 410)
G264	Thread drilling/milling (not TNC 410)
G265	Helical thread drilling/milling (not TNC 410)
G267	Outside thread milling (not TNC 410)

Cycles for milling pockets, studs and slots

G74	Slot milling
G75	Rectangular pocket milling in clockwise direction
G76	Rectangular pocket milling in counterclockwise direction
G77	Circular pocket milling in clockwise direction
G78	Circular pocket milling in counterclockwise direction
G210	Slot milling with reciprocating plunge
G211	Round slot with reciprocating plunge
G212	Rectangular pocket finishing
G213	Rectangular stud finishing
G214	Circular pocket finishing
G215	Circular stud finishing

Cycles for creating point patterns

G220	Circular pattern
G221	Linear pattern

SL Cycles, group 1

G37	Contour geometry, list of subcontour program numbers
G56	Pilot drilling
G57	Rough-out
G58	Contour milling in clockwise direction (finishing)
G59	Contour milling, counterclockwise (finishing)

SL Cycles Group 2 (not TNC 410)

G37	Contour geometry, list of subcontour program numbers
G120	Contour data (applies to G121 to G124)
G121	Pilot drilling
G122	Rough-out
G123	Floor finishing
G124	Side finishing
G125	Contour train (machining open contour)
G127	Cylinder surface
G128	Cylindrical surface slot

Coordinate transformations

G53	Datum shift in datum table
G54	Datum shift in program
G28	Mirror image
G73	Rotation of the coordinate system
G72	Scaling factor (reduce or enlarge contour)
G80	Tilting the Working Plane (not TNC 410)
G247	Disable datum setting (not TNC 410)

Cycles for multipass milling

G60	Running point tables (not TNC 410)
G230	Multipass milling of plane surfaces
G231	Multipass milling of tilted surfaces

*) Non-modal function

G functions

Special cycles

G04*	Dwell time with F seconds
G36	Oriented spindle stop
G39*	Program call
G62	Tolerance deviation for fast contour milling (not TNC 410)

Define machining plane

G17	Working plane: X/Y; tool axis: Z
G18	Working plane: Z/X; tool axis: Y
G19	Working plane: Y/Z; tool axis: X
G20	Tool axis IV

Dimensions

G90	Absolute dimensions
G91	Incremental dimensions

Unit of measure

G70	Inches (set at start of program)
G71	Millimeters (set at start of program)

Other G functions

G29	Transfer the last nominal position value as a pole (circle center)
G38	Program run STOP
G51*	Next tool number (with central tool file)
G55	Programmable probing function
G79*	Cycle call
G98*	Set label number

*) Non-modal function

Addresses

%	Start of program
%	Program call
#	Datum number with G53
A	Rotation about X axis
B	Rotation about Y axis
C	Rotation about Z axis
D	Q-parameter definitions
DL	Length wear compensation with T
DR	Radius wear compensation with T
E	Tolerance with M112 and M124
F	Feed rate
F	Dwell time with G04
F	Scaling factor with G72
F	Factor for feed-rate reduction F with M103
G	G functions
H	Polar coordinate angle
H	Rotation angle with G73
H	Tolerance angle with M112

Addresses

I	Z coordinate of the circle center/pole
J	Y coordinate of the circle center/pole
K	Z coordinate of the circle center/pole
L	Setting a label number with G98
L	Jump to a label number
L	Tool length with G99
M	M functions
N	Block number
P	Cycle parameters in machining cycles
P	Value or Q parameter in Q-parameter definition
Q	Q parameter
R	Polar coordinate radius
R	Circular radius with G02/G03/G05
R	Rounding radius with G25/G26/G27
R	Tool radius with G99
S	Spindle speed
S	Oriented spindle stop with G36
T	Tool definition with G99
T	Tool call
T	Next tool with G51
U	Axis parallel to X axis
V	Axis parallel to Y axis
W	Axis parallel to Z axis
X	X axis
Y	Y axis
Z	Z axis
*	End of block

Contour cycles

Sequence of program steps for machining with several tools	
List of subcontour programs	G37 P01 ...
Define contour data	G120 Q1 ...
Define/Call **drill** Contour cycle: pilot drilling Cycle call	G121 Q10 ...
Define/Call **roughing mill** Contour cycle: rough-out Cycle call	G122 Q10 ...
Define/Call **finishing mill** Contour cycle: floor finishing Cycle call	G123 Q11 ...
Define/Call **finishing mill** Contour cycle: side finishing Cycle call	G124 Q11 ...
End of main program, return	**M02**
Contour subprograms	G98 ... G98 L0

Radius compensation of the contour subprograms

Contour	Programming sequence of the contour elements	Radius compens.
Inside (pocket)	Clockwise (CW) Counterclockwise (CCW)	G42 (RR) G41 (RL)
Outside (island)	Clockwise (CW) Counterclockwise (CCW)	G41 (RL) G42 (RR)

Coordinate transformations

Coordinate transformation	Activate	Cancel
Datum shift	G54 X+20 Y+30 Z+10	G54 X0 Y0 Z0
Mirror image	G28 X	G28
Rotation	G73 H+45	G73 H+0
Scaling factor	G72 F 0.8	G72 F1
Machining plane	G80 A+10 B+10 C+15	G80

Q-parameter definitions

D	Function
00	Assign
01	Addition
02	Subtraction
03	Multiplication
04	Division
05	Root
06	Sine
07	Cosine
08	Room sum of squares c = √ a2+b2
09	If equal, go to label number
10	If not equal, go to label number
11	If greater than, go to label number
12	If less than, go to label number
13	Angle from c ¦ sin a and c ¦ cos a)
14	Error number
15	Print
19	Assignment PLC

HEIDENHAIN

DR. JOHANNES HEIDENHAIN GmbH
Dr.-Johannes-Heidenhain-Straße 5
83301 Traunreut, Germany
☏ +49 (8669) 31-0
FAX +49 (8669) 5061
E-Mail: info@heidenhain.de

Technical support FAX +49 (8669) 31-1000
E-Mail: service@heidenhain.de
Measuring systems ☏ +49 (8669) 31-3104
E-Mail: service.ms-support@heidenhain.de
TNC support ☏ +49 (8669) 31-3101
E-Mail: service.nc-support@heidenhain.de
NC programming ☏ +49 (8669) 31-3103
E-Mail: service.nc-pgm@heidenhain.de
PLC programming ☏ +49 (8669) 31-3102
E-Mail: service.plc@heidenhain.de
Lathe controls ☏ +49 (711) 952803-0
E-Mail: service.hsf@heidenhain.de

www.heidenhain.de

345 940-22 · 8/2002 · pdf · Printed in Germany · Subject to change without notice